Using Logisim Digital Circuit Simulator for Examining Power Electronics Digital Control Systems

Tsveti Hristov Hranov and Nikolay Lyuboslavov Hinov

Abstract – **Logisim is a program used for digital circuits simulation – gates, flip-flops and latches. This paper presents a simple method, which allows simulation of circuits, designed for the power electronics devices, based on digital control units.**

Keywords – **Logisim, Simulation, Digital Control Circuits, Power Electronics**

I. INTRODUCTION

A variety of software for circuit simulation is available in the internet – both free and commercial. Most of them are strictly specialized for electrical circuits, whilst others are for functional simulation. Logisim is an educational tool for functional design and simulation of digital logic circuits [1]. Unlike other electrical simulators it is not as strict as other similar products – for example when you design your circuits it doesn't require from you to define voltage levels, impedance matching and time delay propagation; instead the focus is on the functional description of the logic elements [2]. The software is intended primary for embedded systems engineers, but it can be successfully used in the power electronics field for evaluations of digital control and protection circuits. The simple interface and workflow makes it suitable for focusing on the working mechanism rather than deciding voltage levels, logic product families, parasitic effects or switching characteristics – it gives you answer to the question whether this circuit is suitable functionally for the application or not.

II. CAPABILITIES AND USER INTERFACE

The software consists of a single java archive (.jar), which bundles all the tools, elements and libraries. This makes it very compact, portable and cross-platform – it is open-source, with special educational intention. The starting screen is similar to the one shown on Fig. 1. The main window is divided in five parts: on top there is the menu bar and the toolbar, on the left is the explorer and attribute plane and on the right is the drawing canvas, on which users draw their schematic project. In order to do that they use the primitives from the explorer menu – gates, plexers, latches, flip-flops and some special elements such as buses, clock sources, transistors, transmission gates, LEDs, buttons, registers, RAM, ROM and others. They can be included in the schematic by

T. Hranov is with the Department of Power Electronics, Faculty of Electronic Engineering and Technologies, Technical University - Sofia, 8 Kliment Ohridski blvd., 1000 Sofia, Bulgaria, e-mail: chranov@tu-sofia.bg

N. Hinov is with the Department of Power Electronics, Faculty of Electronic Engineering and Technologies, Technical University - Sofia, 8 Kliment Ohridski blvd., 1000 Sofia, Bulgaria, e-mail: hinov@tu-sofia.bg

simply dragging the desired element into the canvas. The wiring is done by simply clicking on a port (which is denoted by a small dot on every element) and dragging the wire to the desired destination.
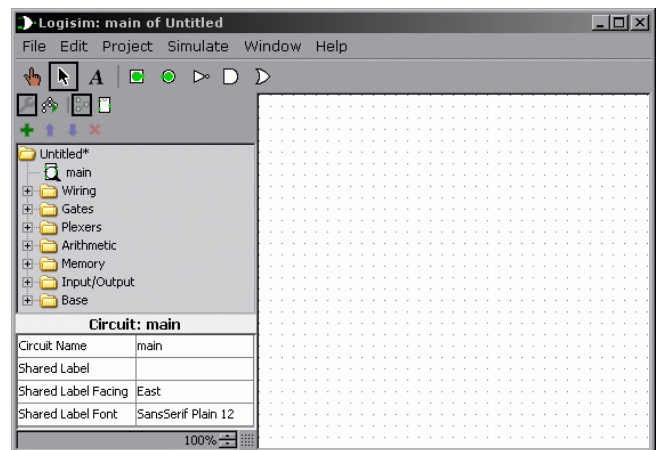

Fig. 1. Logisim main window at start-up

The schematic can be evaluated using the ports (both input and output), probes and clock sources. This is done using the "Poke Tool" (a little hand with index finger on the toolbar) and clicking on the various input ports or clock sources in the schematic. There is support also for automatic clock generation, which can be accessed through the "Simulate" menu, the "Ticks Enabled" option. Tick rate can be set up using "Tick frequency" in the same menu. Also in the same file there can be created several "Sub-circuits" which can be used to make the main schematic more compact and clear to read. This can be done using the "Project" menu, "Add circuit..." option. The circuit appearance can be modified with basic polygon drawing tools using "Modify circuit appearance". Last but not least is a feature to analyze and log circuit behavior, which can be used to synthesize circuit or examine the circuit's functional states. For ease of use elements can be copied, cut and pasted, actions can be undone all via the "Edit" menu. The "File" menu allows for loading, saving, printing and exporting the project in various graphical formats.

III. APPLICATION EXAMPLES

To illustrate the functionality of the software we will discuss two types of circuits used in power electronics: a protection circuit and pulse-drive circuit.

A. Protection circuit

The protection circuits are fine examples of asynchronous digital logic device, which is expected to

react in rare and time-unpredictable moments. Therefore its functional evaluation and testing before design is most critical. For our example we will discuss a circuit which monitors for simultaneous activation of switches in one leg in bridge inverter circuits, such as the one given on Fig. 2. [9, 10]
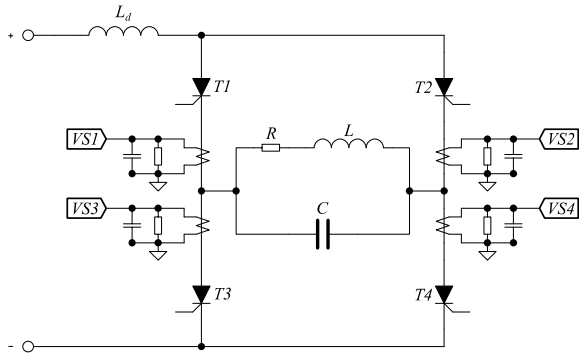

Fig. 2. Thyristor bridge inverter with current transformers

A leg is called the series of two thyristors – in the circuit the two legs are formed by *T1*, *T3* and *T2*, *T4*. A simultaneous firing of those thyristors leads in shorting the supply block, which could lead to a disastrous scenario. The protection circuit must detect this and counteract to it by stopping the supply source. This is done through the current transformers, which create signals for the gates shown on Fig. 3. The resistors and capacitors are there due to the nature of thyristor commutation – since it is not instantaneous, the capacitor gives the required time delay between the pulses [10], in which the current through the thyristors fall to zero in order not to confuse the system. However if the pulse is still there after the time delay has passed this indicates that there is a short in the leg, which should lead to immediate stopping of the supply unit.
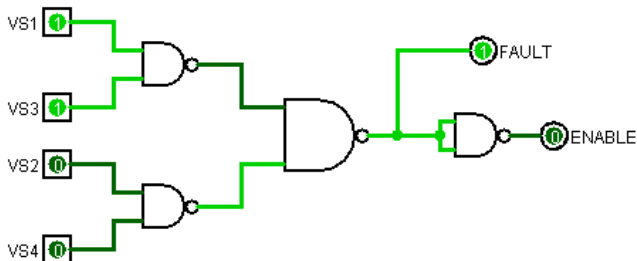

Fig. 3. Bridge Inverter digital protection circuit

The circuit is quite simple, yet effective – the inputs of the two NAND elements are connected to the current sensors for each switch, in a manner that single gate is connected to the two switches in a single leg. The outputs are compared with another NAND gate [3, 4], which will be active when either one of the legs is conducting (that is the power supply is shorted). The function of the schematic can be evaluated with the poke (hand) tool – individual bits can be toggled and the result will be displayed in real-time with the different signals in the wires.

B. Pulse-drive circuit

The control circuit of an AC-mains controlled rectifier is a fine example of a complex synchronous circuit. It has to

be synchronized with the mains frequency and have a stable clock in order to precisely generate a proper delay angle. The example circuit is shown on Fig. 4 [7, 8]. The thyristors are fired during the corresponding half-waves after a defined time delay, which multiplied by the angular frequency ω has the dimensions of an angle – delay angle. In order to do this, the control circuit must detect the zero-crossing, and properly time the moment for the firing impulse [6, 8]. This is usually done with a comparator and a reference voltage, which are not shown here for clarity.
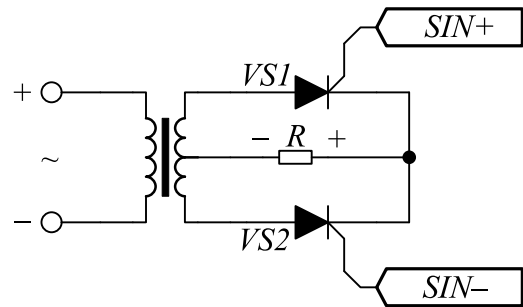

Fig. 4. Full wave controlled rectifier with thyristors and center-tap transformer

The idea is to use the built-in mechanisms of Logisim for clock generation – the clock element. This block changes state ("0" and "1") at predefined "Ticks" – which can be set via the attribute plane ("High duration" and "Low duration"). The asymmetry in the durations allows even generation of Pulse-width modulation (PWM). The digital control circuit is shown on Fig. 6. The mains frequency is simulated via one clock element and the digital clock by another [5]. The main element is a four bit counter, which is triggered to count on a rising edge (for the positive half-wave) and on a falling (for the negative half-wave). It is encapsulated in a sub-circuit and shown as a single element here. When the counter is overflowing, the AND gate is generating a firing pulse for monovibrator and via the feedback is resetting the T-triggers, thus disabling the further counting. The conduction angle can be varied via the frequency of the clock pulses on the counter's counting input. This frequency should be always a ratio between the clock frequency, and this ratio should be greater in order the counter to overflow before the other half-wave takes place.

Another feature that is interesting to illustrate is the ability to simulate the designed circuit and log the inputs and outputs. This is done via the menu "Simulate", option "Logging". Then a window with three tabs pops up (Fig. 5) – in the first tab "Selection" you select which signal states you want to log. You can even log the state of buses and change the view radix. In the second tab "Table" you can observe the states in real-time as soon as you start the simulation.

The third tab "File" allows you to choose the file in which you want to log the states. You can start and stop the file logging anytime using the button above the file selection dialog. The "Include Header Line" option puts a line with the signal names – quite handy with many signals. The simulation can be started and stopped via the menu "Simulation", option "Ticks Enabled". You can observe the signal values in real-time and even if there are input ports

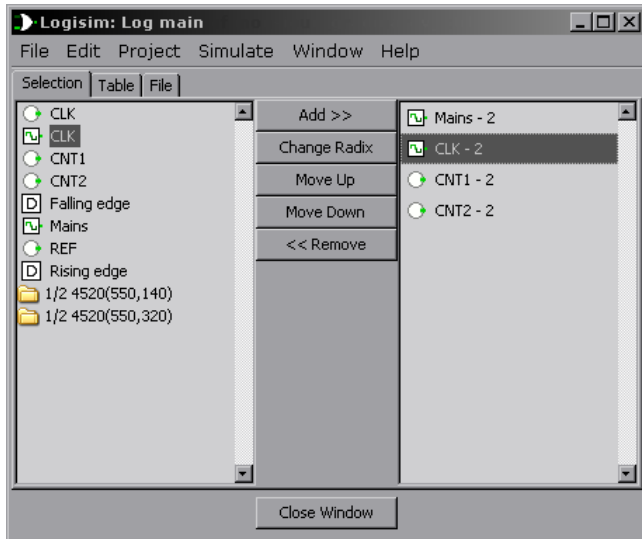or buttons in the schematic they can be clicked to simulate an asynchronous event.

Fig. 5. Simulation logging window

The export file format is plain text, so it is quite easy to create tables and do Boolean and arithmetic operations on the signals. The logged signals can be imported another software and the user can generate the waveforms of the various signals. For our example we will use the software "Graph", which in-depth discussion is not placed here. In order to accomplish this we do some data formatting using spreadsheet software (sorting into columns, formatting the values etc.) and import the values as point series, where x is the time and y is the logical level. An example waveform is given on Fig. 7. For clarity is added also the mains sinusoid, to illustrate the working principle of the controlled rectifier. This assists also when you need to create waveforms and function diagrams for fine preview – for example posters, journals and books. The exported graphics are of vector type, which are best known for the precision and high resolutions capability without distortion like the raster images. The built-in tool for function analysis and evaluation can be used to precisely trace the values with respect to time, to see for example where multiple blocks switch to.
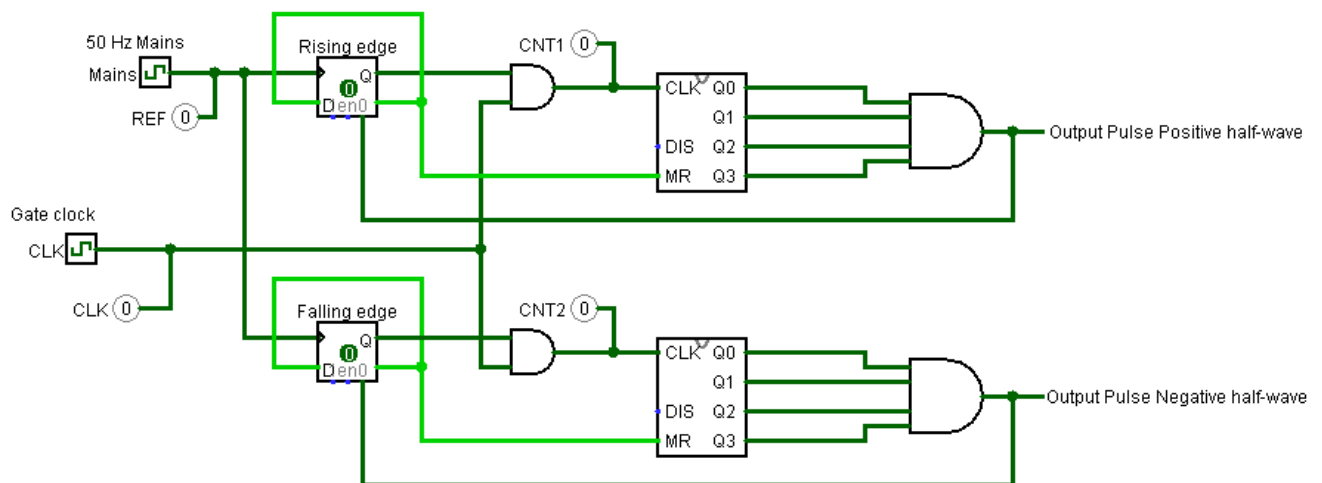
IV. CONCLUSION

Logisim is an excellent education tool which can be used to study and simulate digital circuits. Its simplicity allows the designers to focus on the functional analysis, rather than bothering in such early stages with details like operating voltages, timings, impedances and so on. This cuts the time to evaluate the performance of the solution and if there are required modifications, they can be easily done without this to lead to entire redesign of previous blocks. As a downside it can be pointed out that this version doesn't support direct export of the waveforms – it must be used in conjunction with other software. In our opinion this software is not so familiar in Bulgaria, whose introduction is one of the main goals of this paper. It is intended only for digital simulation and so it lacks elements like capacitors and inductors, which are usually used for setting time-constants. Last but not least thing to note is that this software is completely open-source and available to the public to be used and modified freely, which benefits a lot the academic institutions.

REFERENCES

[1] http://www.cburch.com/logisim/
[2] https://mixedsignal.wordpress.com/2011/02/04/top-ten-circuit-simulators/
[3] G. S. Mihov. *Digital electronics*, Technical University Sofia academic press, ISBN 954-438-449-9, 2005. (in Bulgarian).
[4] G. S. Mihov. *Guide for seminar exercises in digital electronics*, Technical University Sofia academic press, ISBN 954-438-272-0, 2004. (in Bulgarian).
[5] Wakerly John. *Digital Design: Principles and Practices*, Prentice Hall Xilinx Design Series, 4th Edition, 2005.
[6] Mohan, Ned; Undeland, Tore M.; Robbins, William P., *Power Electronics - Converters, Applications, and Design, (3rd Edition)*, © 2003 John Wiley & Sons.
[7] Rashid M. H., *Power Electronics handbook: devices, circuits, and applications*, Academic Press, 2007.
[8] N. Gradinarov, E. Popov and others, *Guide for laboratory exercises in Power Electronics*, Technical University Sofia academic press, ISBN 954-438-420-0, 2007. (in Bulgarian).
[9] Erickson R. W., D. Maksimovic, *Fundamentals of Power Electronics, Second Edition*, Kluwer Academic Publishers Group, 2001.
[10] N. Nachev, G. Malev, *Power Electronics*, Technica, Sofia, 1979. (in Bulgarian).
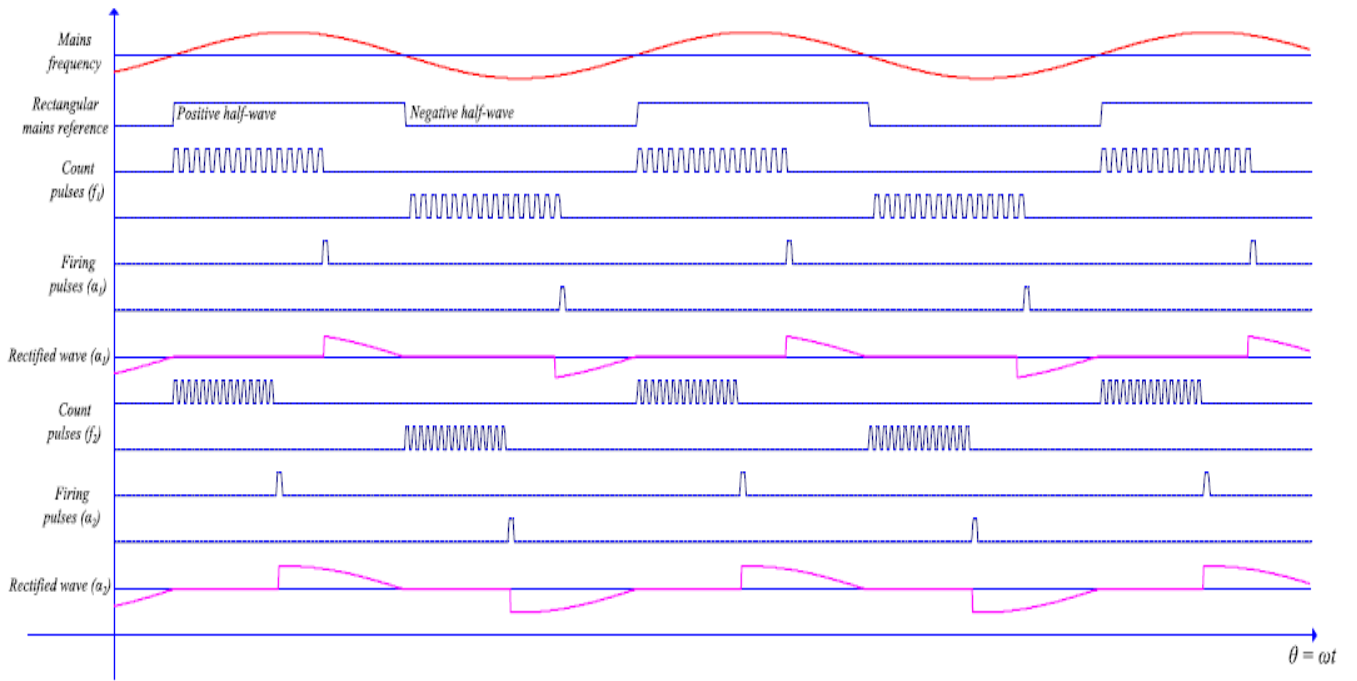
Fig. 6. AC-mains controlled rectifier pulse generating circuit

Fig. 7. Waveforms of the controlled rectifier