Implementation of Electronic Design Automation software tool in the learning process

Trpimir Alajbeg, Mladen Sokele
Zagreb University of Applied Sciences, Department of Electrical Engineering, Zagreb, Croatia
trpimir.alajbeg@tvz.hr, mladen.sokele@tvz.hr

Abstract - Design of analog and digital electronic circuits is an integral part of the electrical engineering curriculum. Software tools, which enable design, simulation, measuring and circuit analysis as well as PCB (Printed Circuit Board) design, are called EDA (Electronic Design Automation). The paper describes the introduction of EDA software in the first semester of the professional study of electrical engineering at the Zagreb University of Applied Sciences within course Personal Computers in Electrical Engineering. Examples of laboratory exercises related to electrical circuit diagrams, use of component libraries, work in standard laboratory environment (power supplies, function generators, voltmeters, ammeters, oscilloscopes, etc.), circuit analysis in a DC and AC mode as well as in time and frequency domain are described. Focus is on the optimal implementation of EDA tool NI Multisim and its integration with the existing learning management system Moodle as a platform for exercise learning and knowledge assessment.

Keywords: Electronic Design Automation, EDA, Multisim, circuit analysis, LMS, learning and knowledge assessment

I. INTRODUCTION

The design of analog and digital electronic circuits, the manufacture of a prototype, testing and after that device production are common tasks of electrical engineering. Design automation of electronic systems is the engineering science that derives software and hardware tools for the design of integrated circuits and systems based on abstraction, design methodologies, and software implementations of sophisticated algorithms for verification and synthesis [1]. Software tools that enable the design of electronic circuits, circuit analysis and measurements as well as preparation for printed circuit boards (PCB) are generally referred to as EDA (Electronic Design Automation) software. In addition, EDA has the ability to simulate and work with programmable digital components such as microcontrollers. Circuit analysis in the design phase is important for emulation of circumstances that may affect the functionality, layout and final dimensions of the product (e.g. the width of the printed circuit, component spacing, parasitic capacitance and frequency range, thermal stresses and cooling, total power consumption) [2].

EDA tools have enabled the integrated circuit industry to sustain exponentially increasing product complexity growth to the present day, while maintaining consistent product development timeline and costs. EDA started as a field in the 1960s when researchers of some leading academic and industrial labs conceived the first computer-aided design (CAD) tools for supporting engineers in the analysis and layout of circuits and boards whose complexity was growing dramatically. State-of-the-art EDA includes features from simulation to physical design, from formal verification to logic optimization as well as of its evolution into system level design, nanotechnology, and synthetic biology [1]. Significant features of EDA are [2]:

- Wide component library including real manufacturers’ components;
- Ability to create new virtual components by user;
- Circuit analysis (DC and AC mode, transient analysis, transfer function analysis, temperature analysis, etc.);
- Creating parts of the technical documentation (parts list, electrical scheme, PCB (Printed Circuit Board) layout, connection plans, etc.);
- Netlist creating (standard data set for external PCB module or other modules).

Introduction to EDA software is curriculum of Personal Computers in Electrical Engineering (PCEE) course. PCEE is being held in the first semester of the professional study of electrical engineering at the Zagreb University of Applied Sciences. Laboratory practicum for PCEE is held on personal computers only [3]. Due to this fact as well as the intention to bring closer work in standard laboratory environment as soon as possible to students of the first semester, it is decided to introduce EDA software tool in PCEE curriculum.

The software used is National Instruments Multisim. For learning process and EDA knowledge/skill assessment, the LMS (Learning Management System) Moodle is used as a platform for all laboratory exercises. Circuit analysis using Multisim allows the user to: Observe the circuit behavior before the actual manufacturing; Use ideal components to isolate design and circuit limitations; Make measurements that are hard to make in the real circuit (because they might damage the circuit, they are affected by electric noise; not possible because of a lack of proper measurement equipment); Perform repeated simulations with parametric values for a component; Observe temperature dependence of the circuit behavior and Observe circuit behavior under parasitic elements due to real components [4].
The following section (Section II) describes the introduction of EDA software with examples of laboratory exercises related to electrical circuit diagrams and circuit analysis. Section III describes the implementation of LMS Moodle platform for exercise learning and knowledge assessment for EDA software tool as well as benefits of its integration.

II. EXAMPLES OF LABORATORY EXERCISES FOR EDA

Design process of an electrical device in the EDA begins with drawing the circuit schematic diagram on the EDA working surface and then follows the circuit analysis. Therefore, laboratory exercises are designed to gradually cover required knowledge needed for successful use of EDA tools: Introduction in Multisim workspace, Measuring instruments usage, Circuit DC analysis, Circuit AC analysis at single frequency, Time domain and Transient analysis and Frequency domain analysis [2].

It is worth to mention that above-stated PCEE laboratory exercises rely on topics of Fundamentals of Electrical Engineering course, which is held in the first semester, too. This enables students to examine acquired knowledge in a simulated environment.

A. Exercise: Introduction in Multisim workspace

Aim of this exercise is to familiarize students with the software tool interface, organization of toolbars, component libraries and their use, to draw a simple circuit diagram as well as to start simulation process. Steps are as follows:

- Understanding the interface, menu, toolbars and workspace;
- Understanding the component library, to differentiate real from virtual components, properly selecting and adjusting component values;
- Understanding the component and measuring instruments placing and wiring on the workspace;
- Clarifying the role of the GND common point (Ground terminal);
- Starting the simulation process to obtain results.

![Fig. 1 Schematic diagram in Exercise A: Introduction in Multisim workspace][2]

Circuit chosen for the exercise has LED (Light Emitting Diode) lights and a switch as simple animated components convenient for the first student work in Multisim (students can immediately see when the circuit is working or not working) – Fig. 1.

B. Measuring instruments usage

The aim of the following exercise is to learn how to work with standard laboratory equipment, such as power supplies, voltmeters, ammeters and oscilloscope. Virtual instruments or simulation of real instruments (Fig. 2) can be selected. All measurements are done in Interactive simulation mode.

![Figure 2. Simulation of real oscilloscope in Multisim used in Exercise B][2]

Circuit in Fig. 3 is used as the first example in this exercise. DC power source is used to make easy understandable initial use of the virtual (simulated) laboratory instruments, particularly to clarify the multimeter input impedance.

![Figure 3. Circuit used in Exercise B][2]

In the second example in this exercise, AC power source is used in rectifier circuit. As measurement instruments, an oscilloscope together with ammeters and voltmeters are used (Fig. 4).

![Figure 4. Rectifier circuit used in Exercise B][2]

C. Circuit DC analysis

DC Operating point simulation is used for static analysis of circuits suitable for circuits with nonlinear
components (such as diodes and transistors). Results of this simulation is given in form of table (Fig. 5) consisted of chosen loop current and node voltages regarding to the reference point GND. This simulation interactively illustrate students all voltage and current values as well as their polarity / direction at all elements of circuit, which is useful for getting the practical understandings before work in laboratory practicum with real circuits.

Circuit used in Exercise C is presented in Fig. 6. The LED lights are used as indicators for proper operation of the circuit. Multimeters are used for comparison with results of DC operating point simulation. In this simulation, voltage or current probes can be used (see results indicated with “PR” suffix in Fig. 5 and their position on schematic diagram in Fig. 6).

Figure 5. Results of DC operating point simulation for circuit used in Exercise C

<table>
<thead>
<tr>
<th>Variable</th>
<th>Operating point value</th>
</tr>
</thead>
<tbody>
<tr>
<td>I(LED1) / (PR1)</td>
<td>6.3894 m</td>
</tr>
<tr>
<td>I(R1) / (PR1)</td>
<td>1.3156 m</td>
</tr>
<tr>
<td>V(V1) / (PR1)</td>
<td>12.2354</td>
</tr>
<tr>
<td>V0</td>
<td>12.20000</td>
</tr>
</tbody>
</table>

Figure 6. Circuit used in Exercise C

D. Circuit AC analysis at single frequency

AC analysis were performed on circuits with alternating current power sources. Types of probes used for the analysis are current probe, voltage probe and differential voltage probe. Output of AC analysis at single frequency can be in complex number form or in phasor form with magnitude and phase (Fig. 7).

Figure 7. Results of AC analysis at single frequency

<table>
<thead>
<tr>
<th>Variable</th>
<th>Frequency (Hz)</th>
<th>Magnitude</th>
<th>Phase (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>B(LED1)</td>
<td>60</td>
<td>36.88035 m</td>
<td>-1.02329</td>
</tr>
<tr>
<td>F(R1)</td>
<td>105</td>
<td>17.85714 m</td>
<td>0.000000e+00</td>
</tr>
<tr>
<td>F(C1)</td>
<td>105</td>
<td>31.41593 m</td>
<td>0.000000</td>
</tr>
<tr>
<td>F(L1)</td>
<td>105</td>
<td>63.66198 m</td>
<td>-90.00000</td>
</tr>
<tr>
<td>F(V1)</td>
<td>105</td>
<td>20.00000</td>
<td>0.000000e+00</td>
</tr>
</tbody>
</table>

Figure 8. Circuit used in Exercise D

E. Time domain and Transient analysis

In this exercise, the current and voltage transient phenomena in time domain are analyzed for a RC circuit shown in Fig. 9. The function generator is used to generate a symmetric bipolar rectangular voltage at selected frequency. Resulting transient analysis is shown in Fig. 10.

Figure 9. Circuit used in Exercise E

F. Frequency domain analysis

The frequency domain analysis is performed on circuit from Exercise E (Fig. 9). The function generator is used to generate a sine wave in range of frequencies (AC sweep).
Resulting frequency domain analysis for RC circuit is shown in Fig. 11.

![Fig. 11. Frequency domain analysis shows magnitude and phase results depending on frequency](image)

III. INTEGRATION WITH LMS MOODLE AND ITS BENEFITS

The integration of EDA with the existing LMS, where all the other exercises and knowledge/skill assessment are conducted, was realized through the following aspects:

- Exercise attendance record;
- Step-by-step instructions for exercise (circuit diagrams and textual instructions with relevant explanations) and expected result of simulations;
- Tasks for homework exercise, deliberately without detailed instructions but with the expected results;
- Assessment system, which consists of acceptance of numerical measurement results of exam in input forms (shown in Fig. 12), comparison with the correct solution with accepted tolerance, automatic grade of student responses and administration of overall student achievement. In addition, backup for subsequent analysis are enabled via upload of student Multisim (.ms14) files.

![Fig. 12. Example of test task for knowledge/skill assessment](image)

The test is time-limited. Overall duration is set in LMS, so after the expiration of the time the test automatically closes [5].

The statistical analysis of the LMS assessment records can be used to track students’ success and their most common mistakes. Two colloquia are prepared: Test EDA-1 and Test EDA-2. Each of them consists of three tasks covering different simulations of electrical circuits.

To check the correctness of the solution, reading of certain values from EDA analysis and multimeters from the set positions are required. Obtained values in the EDA need to be entered by the students in the LMS exam form. Afterwards, the LMS conduct an automatic assessment and administration of the exam [6]. In the first year of implementation, the Test EDA-1 was attended by 201 students, of whom 72% solved all 3 tasks correctly. Test EDA-2 was attended by 150 students of whom 62% answered all 3 tasks correctly. For the first and second test, approximately 2.3 correct assignments was average per student. An overview of the most common mistakes is given in Fig 13. Note: the typos refers to numeric digit substitution (e.g. 63 instead of 36).

Subsequent analysis of submitted solutions uploaded by students (i.e. Multisim files) found that the incorrect wired circuits, not properly adjusted component values and wrongly conducted measurements were source for **Significantly wrong values** (See Fig. 13). The percentage drop of **Empty Answer** and **Wrong Polarity** in Test EDA-2 indicate that students got better prepared for the test that was held a month after the first.

![Fig. 13. Overview of the most common mistakes based on statistical analysis of the LMS assessment records](image)

IV. CONCLUSION

The introduction of EDA software in the learning process of the professional study of electrical engineering at the Zagreb University of Applied Sciences through laboratory exercises is described. Laboratory exercises present complex form of class delivery because students are not only required to be familiar with the theory behind the particular exercise subject, but they also have to handle with software tool and understand laboratory equipment characteristics and their usage limitations. Therefore, sequence of exercise topics is designed to gradually cover required knowledge needed for successful use of EDA tools: **Introduction in Multisim workspace**, **Measuring instruments usage**, **Circuit DC analysis**, **Circuit AC analysis at single frequency**, **Time domain and Transient analysis** and **Frequency domain analysis**. For each exercise, appropriate circuits are designed and step-by-step learning instructions for their design are implemented in LMS. Acquired knowledge and skill assessment with automatic grade of student responses is implemented via LMS, too. It is shown that EDA-LMS integration enables detection of most frequent student
errors, which gives guidelines for improving teaching process. Developed exercises are initial steps of EDA Multisim usage. More advanced options, such as working with digital circuits, microcontrollers and PCB design, are out of the scope of PCEE course and represent curriculum for higher semester courses.

REFERENCES