RESEARCH PAPER | Engineering | Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

Altium: A Fast Schematic Designer

KEYWORDS: Altium , OrCAD , PCB(Printed Circuit Board), FPGA (Field-Programmable gate array), PCB Library, Schematic Library , topology placement and geometrical placement.

Mr. Sanket Mehta	Mr. Jagrat Mehta
PG Student, EC, CSPIT, Anand	Asst. Professor, EC, CSPIT, Anand
Mrs. Arpita Patel	**Mr. Karan Jasani**
Asst. Professor, EC, CSPIT, Anand	Asst. Professor, EC, CSPIT, Anand

ABSTRACT This paper presents an efficient and fast schematic generator of a particular block diagram or circuit. Schematics are generated in the sequence of block-diagram, topology placement and geometrical placement. The high readability and compactness of the schematics help the designers make quick design decisions by providing good signal flow. The aim of this paper is to easily design schematics in Altium for any component, circuit like FPGA etc. It's main task is to make PCB Library and Schematics Library. The final results show that Altium produces more readable and compact schematics than well-known commercial tools.

I. INTRODUCTION

In the past, to make schematics for any printed circuit board was not an easy task. The increase of chip complexity has necessitated the designers simplify their design in Altium rather than OrCAD. The Altium designs can reduce the design time and cost, greatly. Schematics can make through many designers like OrCAD, Multisim etc. The making of Schematics in OrCAD is hard and very complex compare with Altium. The Features and Format for making schematics is nearly similar for both tools Altium and OrCAD. Therefore, it is difficult to confirm or analyze the final schematics just as the designers intension. To find a component Library in any schematic designer is a main task. Altium has most number of component libraries compare to other designer tools. If we fail to find component library, the time consuming and tedious redesign process is required. As a result, the design time becomes long and design cost will be increased. In Altium, if designer fails to find component than also it is easy to make new component.

It is possible to get error in the circuit that will use for making the schematic. To reduce the design time, it is necessary to analyze the intermediate results of circuit using other tools and to correct the errors in an early design stage. When structure is different from the designer's intension, Altium must provide easy modification of particular Circuit.

In order to follow this flow, designer will generate schematics after checking circuit or block diagram of particular system and then translate its database to that of available component library.

Another reason to make a paper is that students are very enthusiastic about designing and fabrication Printed Circuit Board (PCB). To design and fabricate any PCB, students have to make Schematics about particular system. When a student designs and fabricates a PC board, the students knows he is doing what an engineer at a company would do. Most of students want to fabricate boards to make their project look more professional. Usually, students must be convinced not to fabricate boards as a part of projects because there is not enough time in a semester.

II.SCHEMATICS

A schematic diagram is a delineation of an entire system based on block diagram or circuit. A schematic normally eliminate all the details that are not related to the schematic information. For example, a map of subway may represent a subway station for a rider with a dot; the dot doesn't favor the actual station but it gives the information to viewer. In schematic diagram, the elements are arranged like it can be easily understand by viewer.

In electrical and electronic industry, a schematic diagram is often used to describe the design of equipment. Schematics for electronic circuits are prepared by designers using EDA (Electronic Design Automation) tools called schematic capture tools or schematic entry tools. These tools go beyond simple drawing of devices and connections. Usually they are integrated into the whole IC design flow and linked to other EDA tools for verification and simulation of the circuit under design.

Miscellaneous Software used to make effective schematics listed Below :

- OrCAD
- Altium
- CADSTAR etc.

A. ALTIUM

In Past First Design tools founded in 1985 name was "Protel" , launching the company's first product later that same year – a DOC based Printed Circuit Board(PCB) layout and design tool[6]. The company continued to develop and release new versions of this design tool, following Protel for Windows, the world's first Microsoft Windows-based PCB design system. After some time this name changed to "Altium". Altium was listed to assist in funding of strategic technology development and acquisition.
Altium is a Fast and Efficient schematic designer tool compare to other tools.

B. OrCAD

The name OrCAD is a portmanteau reflecting the company and its software's origins: Oregon+ CAD. OrCAD is a proprietary software tool suite used primarily for electronic design automation[7]. The software is used mainly by electronic design engineers and electronic technicians to create electronic schematics and electronic prints for manufacturing printed circuit board. OrCAD EE Designer

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

plus include OrCAD Capture and PSpice A/D and Advanced Analysis. OrCAD a comprehensive solution for entering, modifying and verifying complex system design.

OrCAD is probably one of the oldest running suits of circuit design and simulation software. It's one of the industry standard tools for professional designers and electronic engineers.

C. Difference B/w ALTIUM And OrCAD
- Altium is a bit easy to use more than OrCAD for beginner.
- Altium has most number of component library compare to OrCAD.
- Simulation can do easily in Altium rather than OrCAD.

Some Designers uses Altium due to several important reasons as enlisted below:
- Optimum ratio between Performance
- Time Management
- Excellent Schematic Browser
- 3D Visualization and output

a.Optimum ratio between Performance:
You can design any board in Altium Designers. For advanced boards (e.g. High Speed Design) the PCB Layout process may take a little bit longer time in Altium when an OrCAD it takes a few days or weeks more.

b. Time Management:
In Altium Designer, once everything is set up Correctly, it just takes a few clicks where in OrCAD, it will takes a lots of time.

c. Excellent Schematic Browser:
You always want to be sure the schematic is correct .
Altium Designer provides an excellent schematic browser which allows you easily check connection for every pin which is present in your schematic.

d. 3D Visualization and output:
Altium Designer has 3D features where OrCAD Doesn't have 3D features. They can see the product weeks or months before it's physically on their table. Visualization is used to check component placement and it's especially useful when board is being designed for a bigger system.

As describe all about Altium and OrCAD, I suggest to use Altium rather than OrCAD for better performance.

PROCEDURE TO MAKE SCHEMATIC IN ALTIUM
Three Basic Steps:
1. Block-diagram
2. Topology placement and
3. Geometrical placement

I. Block Diagram/Circuit :
A block diagram or circuit is a mandatory for making any Schematics. Designer must have circuit or block diagram of particular system. Here, Designer has a "Flashing LED Circuit"

General Description :
This circuit uses the 555 timer in an Astable operating mode which generates a continuous output via Pin 3 in the form of a square wave. This turns the LED (D1) on and off. The speed at which the LED (D1) is turned on and off is set by the values of R1 and R2.

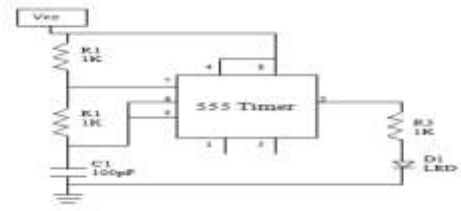

Figure 1. Flashing LED Circuit

Normal design procedure is to simulate the circuit to see if it passes all specifications. If the simulations indicate adequate performance, the circuit is tested in the lab. If the actual circuit does not pass the performance specification, the circuit must be modified on the bread-board and in the simulations. The simulations and testing are then repeated.

Once the circuit simulations match the bread-board performance of the circuit, it is safe to say that the circuit drawn on the computer screen is an adequate representation of the actual circuit and that the circuit works. Moreover, not only are the simulations correct, all connections between components are correct. We know the schematic has the right connections and we will use this schematic to make PCB (Printed Circuit Board).

ii.Topology Placement :
Now Designer knows he has a correct circuit that need to assign component to make schematic. Designer required a knowledge about component like no. of component, component name and placed this component as per circuit as shown in figure[3].

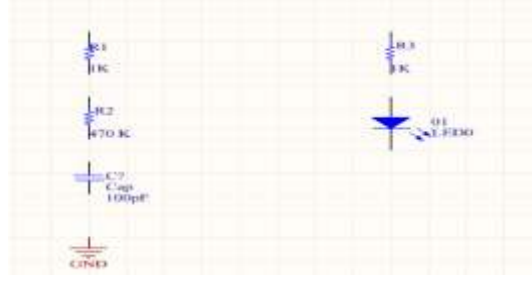

Figure 2. Topological Placement of Component.

iii .Geometrical placement:
When the entire components are placed as per circuit, designer have to connect them by wiring. Here, it can happen that designer get error to find component from library. If designer fail then component library must be design as per requirement. In Altium, it is very rarely happen that designer fails to get component from library.

Here, in given circuit 555 Timer not available in both Altium and OrCAD. The process to make new component library in Altium is very easier than other tools. Entire process of making component library describe below:

There are several steps to make component Library :
- PCB Library
- Schematic Library
- Add Library

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

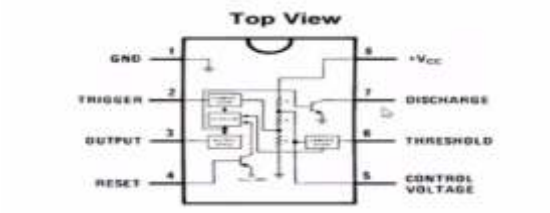

Figure 3. 555 Timer Circuit

1.PCB Library(Footprint):

- **First of all, Designer require some information listed below :**

I. No of pin
II. Spacing between two pin
III. Total Size of Component IC

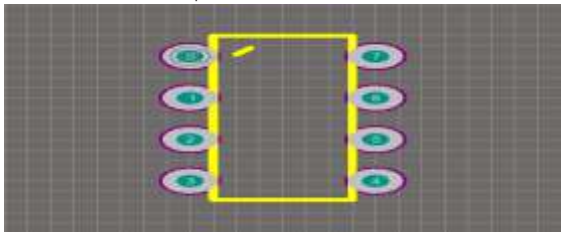

Figure 4. PCB Library of 555 Timer

2.Schematic Library:

- Designer needs to be add footprint in Library so that in future it can use in any pc and for any project.
- Component has to be made same as required for circuit by using tools (Altium).
- Give proper name as per component Datasheet for better readability for designer.
- Save project and your component library is ready to use [3].

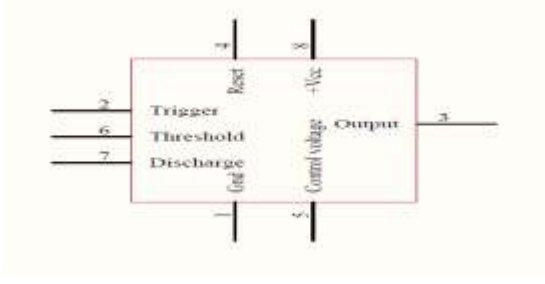

Figure 5. Schematic Library of 555 Timer

3.Add Library
Library → Add Path of your schematic library
Schematic library is ready to use.
Any Designer can use this component library for future.
(Go to the library and Add a path where u save your Schematic)
Now, Designer can able to complete the schematic of circuit [3]

Schematic using Altium:

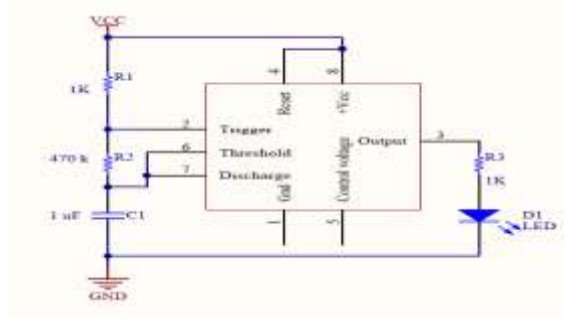

Figure 6. Schematic Using Altium Schematics using Or CAD:

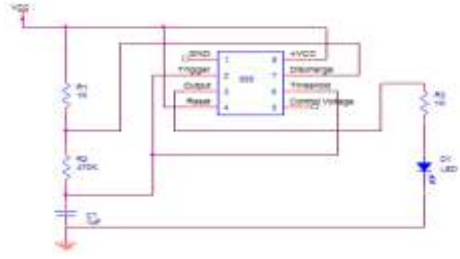

Figure 7. Schematic Using Or CAD[4]

IV. EXPERIMENTAL RESULTS
We tested the performance of Altium by making schematics for several circuits and block diagram. The results of Altium were compared to those of OrCAD from Synopsys.

Table 1 shows the results that designer experienced during making of schematic, where Design time and Simulation refers to the time taken to make design and simulate the entire circuit. Also the 3D Features refer to the enhanced view of schematic. The performance shows the performance of designer. Component library shows library of component present in designer. Complexity stands for the complexity face to design schematic.

Comparison of the Design time, Simulation, Performance and complexity proves that Altium generate far less complex schematic than OrCAD. By compare the component library of both tools, Altium has mostly all components than OrCAD. 3D Features are a supplementary feature exists in Altium that is not available in OrCAD.

Parameter	Altium	OrCAD
Time To Design	Less	More
Simulation	Fast	Slower Than Altium
3D Features	Yes	No
Performance	High	Low
Component Library	Mostly all component	Limited Component
Complexity	Less	More

Table 1. Difference b/w Altium and OrCAD

V. CONCLUSION:
In this Paper, I proposed an efficient schematic generator of Electronics Circuit .The experimental results show that Altium produces more readable and compact schematics than a well-known commercial tool in a real time response. The high readability and compactness of the generated schematics help the designers make early design.

Volume - 5 | Issue - 1 | Jan Special Issue - 2015 | ISSN - 2249-555X

VI. FUTURE ENHANCEMENTS:

Electronic circuits in schools and industry are normally manufactured through the use of PCBs (Printed Circuit Boards). Printed Circuit boards are physical and electrical scaffolds that hold the electronics together and designing them requires a lot finesse and ingenuity. Layouting will be the next step after Creating a schematic circuit to make Printed Circuit Board (PCB) .

3D Features can also be add by using some command which is inbuilt in Altium Designer.

VII. Acknowledgment:

Sometimes words fall short to show gratitude, the same is happening with me during this paper. The immense help and support received from faculty and friends overwhelmed me during the paper.

I am grateful to my friends and faculties who helped me to successfully completion of this paper.

I am highly indebted to for providing me with the necessary information and valuable suggestion and comments on bringing out this paper in the best possible way.

REFERENCE

[1]. Nam-Hoon Kim et al., "RightTopolizer : An Efficient schematic generator for Multilevel Optimization", pp 307-391 | [2]. Marc E. Herniter , " PC Board Design and Fabrication using schematics , PADS-PERFORM, and a Laser Printer." , in Education Conference, pp 111-115 , 1994 | [3]. Nicholas Martin , an electronic designer , Computer Software : Altium Designer , 1985 | [4]. John Darbetaki , Ken and Keith Seymour , computer software : OrCAD , 1985 | [5]. The IEEE website.[online]. Available: http://www.ieee.org | [6]. Altium Website[online]. Available: http://www.altium.com | [7]. OrCAD Website[online]. Available: http://www.orcad.com | [8].Feng et al. , " Model of a TDI Line scan camera and its Electronics" , IEEE, pp 2215-2220