International Journal of Systems Signal Control and Engineering Application
MEDWELL PUBLICATIONS SINCE 2000

Development of Eagle Multi-Layer Printed Circuit Board with CAD and CAM

M.N. Islam, M.S. Alam and M.A.S. Haque
Institute of Electronics, Atomic Energy Research Establishment, Bangladesh Atomic Energy Commission, P.O. Box No. 3787, Savar, Dhaka

Key words: Multi-layer PCB, Schematic Module, Layout Editor, CAD and CAM

Abstract: Development of multi-layer Printed Circuit Board (PCB) by using Eagle has been presented in this project. The Eagle is a CAD tool comprises of Schematic Module, Layout Editor Module, Autorouter Module and CAM Job. The schematic module is for development schematic. After designing the schematic, Electrical Rule Check (ERC) would be performed for electrical errors and consistency check between schematic and board. Thereafter, with the help of this schematic, it needs to create a board in the Layout Editor Module. Then the Digital Rule Check (DRC) has to be operated for reporting any errors in the Board like clearance errors under the set of design rules. Thereafter, 100% finished multi-layer (1-16) board would be created by Autorouter with desired track size and layer. Afterwards, adding ground and mounting holes manufacturing data have to be generated by CAM Job and send the board house for fabrication. Finally, the board house provides multi-layer finished Printed Circuit Boards (PCBs). After fabrication, the board should be assembled and tested successfully. The testing data would be presented for functionality characteristics.

Corresponding Author:
M.N. Islam
Institute of Electronics, Atomic Energy Research Establishment, Bangladesh Atomic Energy Commission, P.O. Box No. 3787, Savar, Dhaka

Page No: 77-80
Volume: 14, Issue 6, 2021
ISSN: 1997-5422
International Journal of Systems Signal Control and Engineering Application

INTRODUCTION

The Printed Circuit Board (PCB) is a laminated sandwich structure of conductive and insulating layers. PCBs have two complementary functions. The first is to affix electronic components in designated locations on the outer layers by means of soldering. The second is to provide reliable electrical connections as well as open circuits between the component's terminals in a controlled manner often referred as PCB design. PCBs can be single-sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer) or multilayer (outer and inner layers of copper, alternating with layers of substrate). Multi-layer PCBs allow for much higher component density, because circuit traces on the inner layers would otherwise take up surface space between components. The rise in popularity of multilayer PCBs with more than two and especially with more than four, copper planes was concurrent with the adoption of Surface Mount Technology (SMT) (Fig. 1). However, multi-layer PCBs make, repair, analysis and field modification of circuits much more difficult[1]. The world market for bare PCBs exceeded \$60.2 billion in 2014[2] and is estimated to reach \$79 billion by 2024[3, 4].

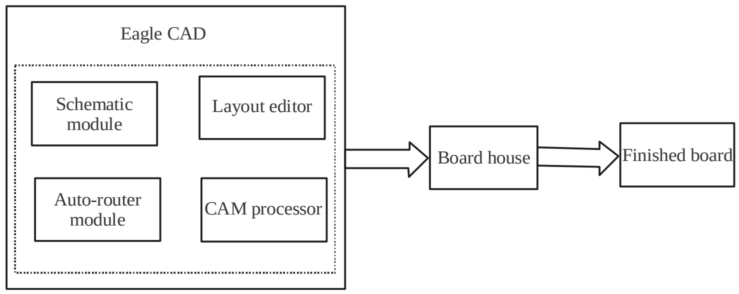

Fig. 1: The CAD tool EAGLE PCB development process diagram

METHODOLOGY

The basic EAGLE software package comes with the Schematic diagram module to layout editor which allows one to design Printed Circuit Boards (PCBs), plus the library editor, autorouter module, the text editor and the CAM processor. With the library editor one can already design packages (footprints), symbols and devices for a schematic. The CAM processor is the program which generates the output data for the production of the PCB (e.g., Gerber or drill files).

If someone has the schematic module one begin by drawing a circuit diagram, then it is possible to generate the associated circuit board at any time with a mouse-click. It needs to perform Electrical Rule Check (ERC) for electrical errors and consistency check between schematic and board. The Digital Rule Check (DRC) has to be operated for reporting any errors in the Board like clearance errors under the set of design rules. EAGLE then changes to the Layout Editor, where the packages are placed next to an empty board connected via airwires (rubber bands). One can route the airwires automatically if one own the Autorouter module. One can choose single nets, group of nets or all nets for the automatic routing pass (AUTO Command). The program will handle various network classes having different track widths and minimum clearances. Schematic and layout are automatically kept consistent by EAGLE (Forward & Back Annotation). Schematic diagrams can consist of up to 99 sheets.

Schematic design: Devices are taken from existing libraries and placed on the drawing area as shown in fig.1. The connecting points (pins) on the devices are then joined by nets (electrical connections). Nets can have any name, and can be assigned to various classes. Power supply voltages are generally connected automatically. In order to document all the supply voltages in the schematic diagram, it is necessary to place at least one so-called supply symbol for each voltage. Schematic diagrams can consist of a number of pages (but not in the Light edition). Nets are connected across all the pages if they have the same name. It is assumed that libraries containing the required components are available (Fig. 2).

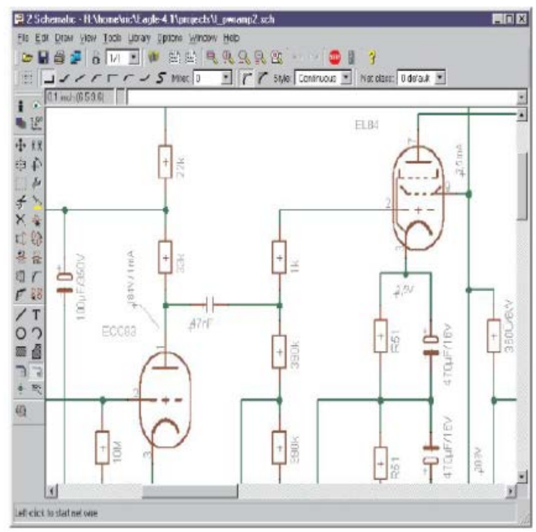

Fig. 2: Schematic editor window

Creating a board: According to Fig. 3 after designing the schematic, click the board icon an empty board is generated, next to which the components are placed, joined together with airwires. Supply pins are connected by those signals which correspond to their name, unless another net is explicitly joined to them. The board is linked to the schematic by the forward and back annotation. Provided that both files are always loaded during editing they are guaranteed to remain consistent. Alterations made in one file are automatically carried out in the other.

Autorouter: When running the autorouter with the AUTO command, the setup menu appears first as shown in Fig. 4. All the necessary settings are made there. This is where one specifies the layers that may be used for

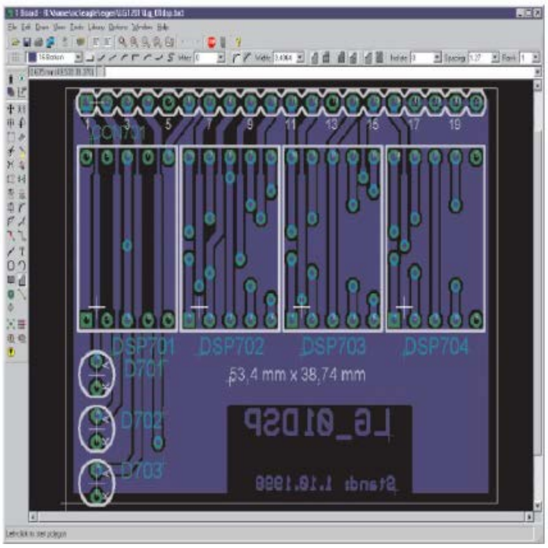

Fig. 3: Layout editor window

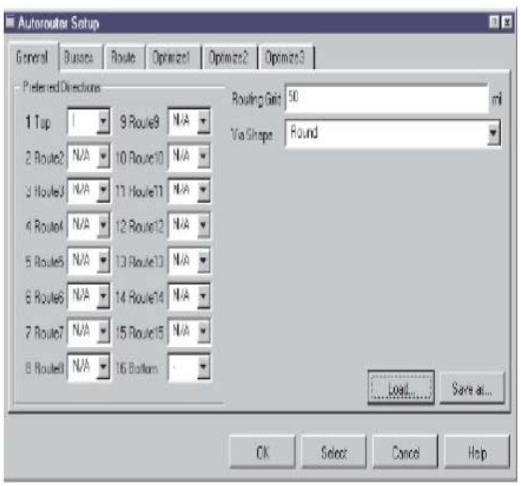

Fig. 4: Autorouter setup (General settings)

routing and preferred directions apply. Click the appropriate combo box with mouse and select the desired value.

CAM job: The CAM processor provides a job mechanism with the aid of which the creation of the output data for a board can be automated as of Fig. 5. The Control Panel's tree view (CAM Jobs branch) lists all jobs and shows a brief description:

- A job is loaded with the file menu of the CAM processor or via the control panel
- Loading a board
- Output parameter
- Defining a job (Fig. 6 and 7)

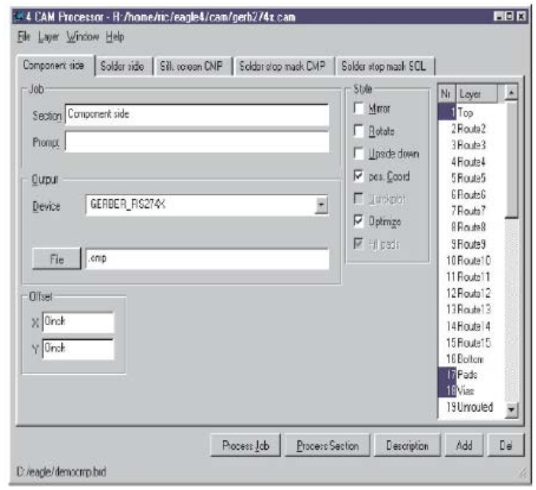

Fig. 5: CAM processor window

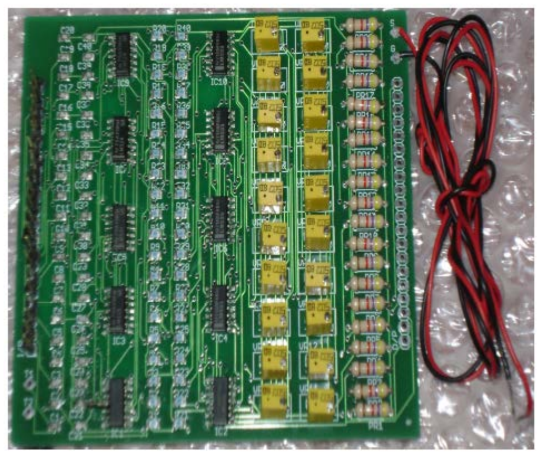

Fig. 6: Eagle 2-layer finished board top (left) after assembling

Fig. 7: Typical PCB manufacturing plant: 5 SMT production lines, 2 DIP assembly lines and >4 complete fabrication lines

- Click add, to add a new section
- Set parameter
- Repeat a and b, if necessary
- Save job with file/save[5, 6]

CONCLUSION

Development of multi-layer Printed Circuit Board (PCB) using Eagle with CAD and CAM has been presented in this study. The CAD tool Eagle comprises of Schematic Diagram Module to Layout Editor, Autorouter Module and the CAM processor. The windows based software package also contains Library Editor, DRC and ERC. This windows program is suitable for development of 1-16 layer 100% finished PCB successfully.

REFERENCES

01. Anonymous, 2014. World PCB production in 2014 estimated at $60.2B. Wikimedia Foundation, San Francisco, California.
02. Anonymous, 2018. Global Printed Circuit Board (PCB) market to witness a CAGR of 3.1% during 2018-2024. Research, Energias Market, Dehradun, India.
03. Anonymous, 2018. Global single sided printed circuit board market- growth, future prospects and competitive analysis and forecast 2018-2023. The Industry Herald, Australia.
04. Anonymous, 2004. EAGLE easily applicable graphical layout editor EAGLE manual, version 4.1. CadSoft Corporation, USA.
05. Nazrul, I. and H. Takahashi, 2007. Nuclear researchers exchange program 2008. University of Tokyo, Department of Nuclear Engineering and Management, Tokyo, Japan.