iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 **Issue:** IV **Month of publication:** April 2023

DOI: https://doi.org/10.22214/ijraset.2023.50923

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

Arduino-Based Embedded System Module

Harshita Sharma[1], Hemlata Prajapat[2], Jayant Yadav[3], Lakshya Bhatra[4], Akash Deo[5]
[1, 2,3,4]Students, [5]Assistant Professor, Dept. of Electrical Engineering, Swami Keshvanand Institute of Technology, Management and Gramothan, Jaipur, Rajasthan, India

Abstract: An embedded system is a special-purpose computer system. They are important now a days as they control numerous common devices we use. It is designed to perform functions, frequently with real-time computing constraints. The project objective is to design a development board that consists of various modules of embedded system applications. Arduino board has allowed numerous engineers to incorporate embedded systems in their designs due to its ease of operation. In this project, PCB is designed on a circuit design software DIPTRACE, i.e. an ECA/CAD software to make schematic design and printed circuit board. This project facilitates the end users by providing different operations of an embedded system on a single development board, like a Gas sensor, Temperature sensor, motor driver, etc.
Keywords: PCB, Embedded system, DIPTRACE, ECA, CAD.

I. INTRODUCTION

The term "embedded system" refers to the union of software and hardware. The dropping price of computing power, along with the falling price of memory, and the capability to design low-cost systems on a single chip, has led to the development and deployment of embedded computing systems in a wide range of operation surroundings. We will use the embedded system design process as a framework for the study of co-design[6]. Hence, an increasingly effort has been devoted to AoI optimization in embedded systems[7]. An Arduino board is preferred because of its simplicity of use for integrating embedded systems into the design of various circuits. Microcontrollers deliver a better outcome since they are computers on single-chip that enable the creation of embedded smart systems, which are widely used today [1]. A Microcontroller is a small computer on a single chip comprising a processor, memory, and programmable input and output peripherals [2]. Consequently, there is an surge in longing for new knowledge among engineers and students also, especially in the areas of prototyping power electronic instruments to save time, ameliorate efficiency, and promote artificial intelligence. Microcontroller topologies are also being created and enhanced to satisfy the demands of design engineers. The knowledge of digital electronics concerning conversion from one base to the other will enable him to program them. Arduino is an open-source prototype platform that abridges the art of designing embedded systems. It comprises a board appertained to as an Arduino board and can be programmed in its Integrated development environment IDE [3]. Circuit prototyping is a crucial step in any electronic design process. Electronic circuit prototypes have typically been constructed on turret boards or breadboards [4, 5], the latter of which is still the most popular prototyping method used today. The industry evolutions have increased the importance of printed circuit boards (PCB) in electronics prototyping [11]. Arduino UNO microcontroller core platform is employed in this project. It can be fluently interfaced with computers, drivers, and stepper motors as well. PCBshave the additional advantage of providing robustness that breadboards cannot achieve[8]. Finally, PCBs can be duplicated with ease, allowing parallel testing of multiple (nearly) identical prototypes. Despite their numerous advantages in the prototyping phase of electronic design processes, usage of PCBs outside professional environments and high end academic research groups is undervalued[9]. With the recent introduction of professional yet easy to use and affordable design suites [10], the design of a PCB is no longer a prohibitive factor. In contrast, production processes for PCBs have become more complex due to environmental limitations and advancing electrical requirements. As a result, the proposed module gives users the convenience of operating and testing a variety of embedded system applications on a single board by fusing several modules together as needed. The details of the module are given in section II.

II. PROPOSED MODULE

The development board has six independent modules that can be connected to Arduino in various ways depending on the needs of the user. Here, DIPTRACE software, an ECA/CAD programme used to create printed circuit boards and schematic designs, is utilised to create the modules. One can quickly create any schematics and then change them to other formats like PCB. . It is a piece of software for circuit design that increases circuit design productivity. Diptrace provides user with the advantage of immediately checking for mistakes while enabling user to swiftly design complex circuits. It enables the user to make use of fundamental and adaptable features like 3D modelling, quick shape-based routing, and broad import and export options. It offers multi-sheet and multi-level hierarchal circuits. The block diagram of the proposed module is shown in Figure (1).

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

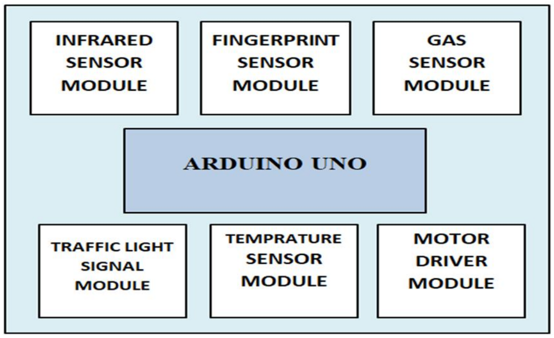


Figure 1: Block diagram of the proposed module

The steps followed in the design of the proposed module are:

1) Open the editor section in the tool
2) Select either components/footprints to be created
3) Create new components
4) Create new footprints
5) Schematic capture
6) Verify the circuits through the simulation procedure
7) Run simulation
8) Conversion of Schematic Layout to PCB

III. SCHEMATIC DESIGNING

Initially, a schematic design of the printed circuit board must be created. The circuit needs to be set up. Making a schematic in advance will be useful as a guide for positioning the traces and all of the components on the PCB board. Additionally, the PCB designing software may be able to import all of the footprints, wires, and components onto the PCB file, making it simple to build circuits overall. The user can connect the pins logically, visually, without wires, or by using the inbuilt net ports on the circuit board in the schematic capture section. It can import and export data from different CAD/EDA applications and formats, convert schematics to PCBs, and annotate simply and be imported/exported from other CAD/EDA software and formats. EasyEDA is a web-based software programme that enables you to design and simulate electronic circuits as well as produce printed circuit board (PCB) layouts. It is used to create the schematic for the proposed module. It is an easy-to-use software application for electronics that is appropriate for both newcomers and experts. EasyEDA offers a drag-and-drop schematic capture interface that makes it simple to position and connect electronic components. The schematic of the proposed module is shown in below figure(2).

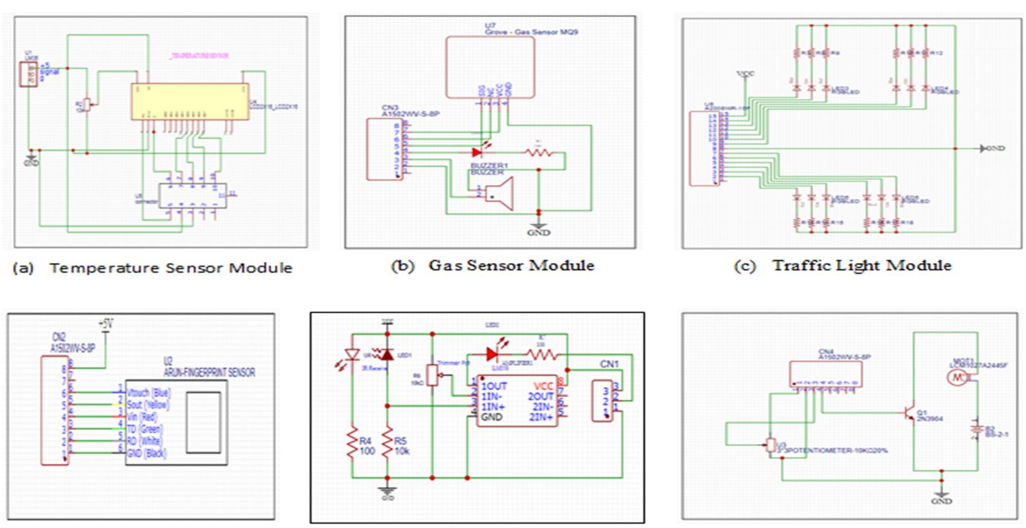

(a) Temperature Sensor Module (b) Gas Sensor Module (c) Traffic Light Module
(d) Fingerprint Sensor Module (e) Infrared Sensor Module (f) DC Motor Module

Figure 2: Schematic Diagram of modules

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue IV Apr 2023- Available at www.ijraset.com

IV. PCB LAYOUT

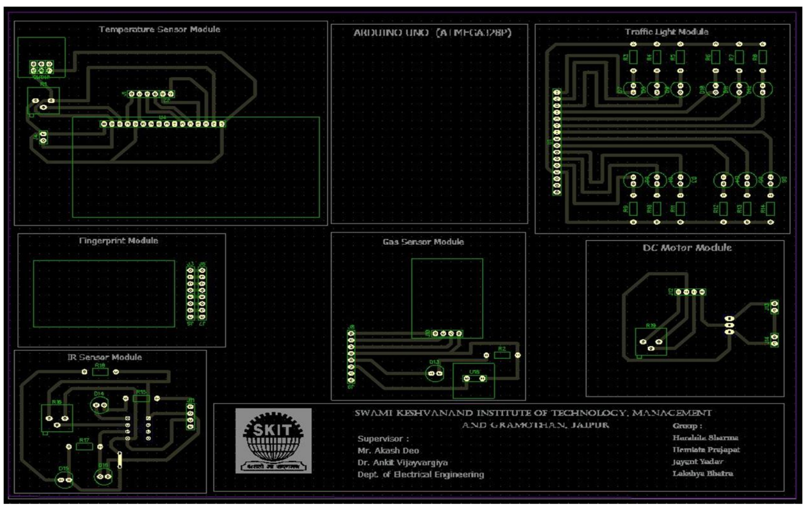
Diptrace is a top-notch board-level design tool that can be used for intelligent placement, routing tools, shape base auto-router, copying blocks, and creative projects. The verification feature is one of the fundamental qualities from which user may determine using high-speed signals, we may swiftly verify the accuracy of the intricate project. PCB layout allows the user to edit and create printed circuit boards. Switching is advantageous since it is difficult to construct courses in PCB layout directly and is necessary when creating complex circuits.

Some basic steps for creating PCB Layout with dip trace :

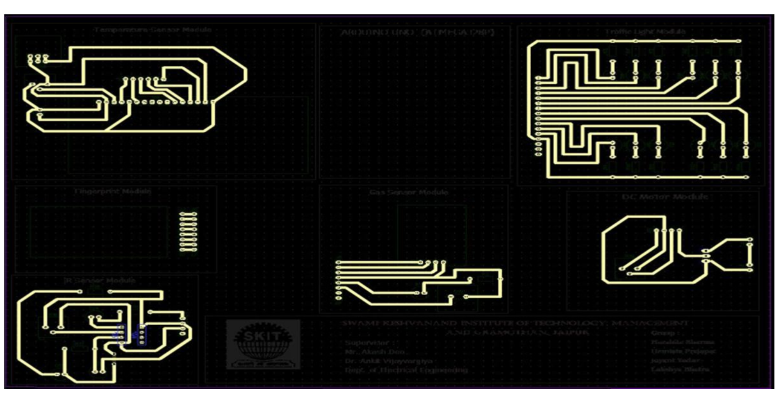
1) Import the schematic
2) Create a new PCB Layout
3) Place components
4) Route traces
5) Define copper layers
6) Modify component placement
7) Make silkscreen

There are various processes involved in moving from schematics to layout, which are listed below:

a) Choose the circuit design, then pick the file, convert PCB, or press Ctrl+B. You can convert the schematics to the PCB layout in this way.
b) You may need to renew your schematics for this at times. Navigate to the "file/renew layouts from the schematics" option in the PCB layout's main menu.



(a) Top view



(b) Bottom view

Figure 3: Layout of PCB

V. PCB DESIGN PROCESS

Designing a Printed Circuit Board (PCB) involves several steps that need to be followed in sequential order.

1) *Determine the System Requirements*: Identity the purpose and functionality of the embedded system to be integrated into the PCB. Determine the required components, the power supply requirements, and the interfaces that need to be included in the design.
2) *Create a Block Diagram*: Develop a block diagram to visualize the main functional blocks of the system and their connections to each other.
3) *Schematic Capture*: Use a schematic capture software tool to design the circuit diagram of the PCB with all the required components and their connections.
4) *Component Placement*: After the schematic capture process, place all the components onto the PCB layout design. Consider the physical size and shape of the components and their placement to each other.
5) *Routing*: Route the traces that connect the components according to the schematic capture design.
6) *Design Rule Check*: After completing the layout design, run a Design Rule Check (DRC) to identify any errors or warnings. This process ensures that the PCB design meets the manufacturing requirements and standards.
7) *Generate Gerber Files*: Once the DRC process is complete and the design is error-free, generate the Gerber files that will be used to manufacture the PCB.
8) *PCB Assembly*: After receiving the manufactured PCB, assemble the components onto the board. Follow the recommended soldering techniques and procedures to ensure the proper functioning of the embedded system.
9) *Testing*: Finally, test the PCB to ensure that it meets the design specifications and functional requirements.

VI. PCB FABRICATION

The PCB manufacturing process as discussed below is split up into 6 phases The 6 phases include the Copper Routing Phase, the Drilling Phase, the Solder Resist Phase, the Though Hole Plating Phase, the Surface Finish Phase, and finally the Component Overlay Phase.

1) *Copper Routing Phase*: The Copper Routing Phase is the only essential phase of the manufacturing process. Two major methods exist for routing copper-clad boards: mechanically through milling, and chemically through etching. In milling price range is high and PCB milling machines require a processing time that is proportional to the complexity of the board being milled. But milling is more environmentally friendly due to the absence of chemicals and the easier reuse of removed copper. The alternative to milling is an etching, which refers to chemically removing undesired copper from the copper-clad board utilizing a corrosive chemical etchant. Etching times range from a few seconds to dozens of minutes. Copper routing using chemical etching requires 4 steps: transferring the artwork to an etch resist, developing the resist, etching the board, and finally stripping the extra resist from the copper.
2) *Transferring artwork to etch resist* : Coating that can resist the etchant must be applied before etching. This photosensitive coating can be applied to a blank board as a film. To transfer the copper layer information to During the etching of copper-clad board, an etchant is used to submerge the board. To protect desired copper areas such as tracks and polygons from the etchant, a the board, it is first printed on cellulose acetate transparencies using a high-resolution laser printer. Once printed, the protective layer is removed from the copper-clad board and placed in an ultraviolet (UV) exposure unit. The transparency is then aligned with the board, and exposed to ultraviolet light to cure the uncovered parts of the photosensitive laminate. Tracks are covered by ink on the transparency and protected from ultraviolet light. Any ultraviolet light source can be used to cure the photosensitive coating from a technical perspective, including sunlight.
3) *Developing the etch resist:* After curing the photosensitive coating above undesired copper, it must be washed off before commencing the etching process. This is done by developing the photosensitive coating in a 1% sodium hydroxide (NaOH) solution at room temperature. After developing, the board is rinsed with water to prevent NaOH from entering the etching solution.
4) *Etching the board*: Any chemical substance, or combination of chemicals, may be used as the etchant to dissolve metallic copper (Cu) while causing no damage to the board's etch resist. Etchants are generally divided into 5 groups: peroxides, metal chlorides, chlorine-based etchants, persulfates, and nitrates. Etchants used in PCB manufacturing are- Due to its affordability and speed of etching, ferric chloride ($FeCl_3$) is one of the most commonly used etchants for PCBs. As a ready-to-use solution, it is offered commercially. Other etchants, such as sodium persulfate ($Na_2S_2O_8$), hydrochloric acid, and hydrogen peroxide (HCl and H_2O_2) are also used.

5) *Stripping the etch resist:* After the etching process is finished, the etch resist must be stripped off to expose the copper tracks on the board so they can be treated with a surface finish. . The board is first cleaned with acetone to dissolve the coating before being rinsed with water to remove the tainted acetone in order to remove the etch resist
Following the completion of the aforementioned steps, drilling and component soldering are carried out.

VII. 3-D VIEW

A 3D view of a printed circuit board (PCB) provides a realistic and detailed representation of the physical layout and components of the PCB. It shows the position and orientation of each component on the PCB. This allows for visual inspection of the placement of components, ensuring that they are correctly positioned and spaced for the proper functioning of the circuit.
The 3D view displays the layer stacking of the PCB, showing the different layers of the PCB and their location with each other: This view also shows the routing of the traces, indicating the paths and connections between the components. This allows for a quick and accurate visual check of the traces, ensuring that they are properly routed and spaced for optimal performance.
It also displays the mechanical dimensions of the PCB, including its size, shape, and mounting holes. This enables the designer to check that the PCB fits within the mechanical constraints of the device for which it is designed.

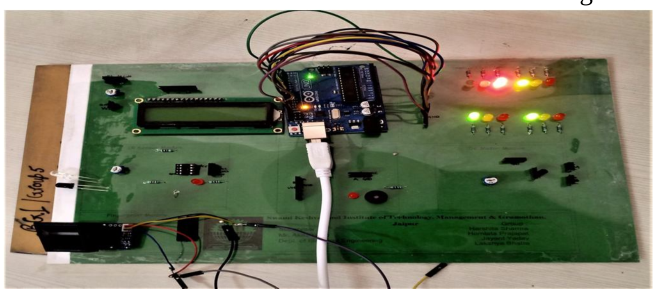
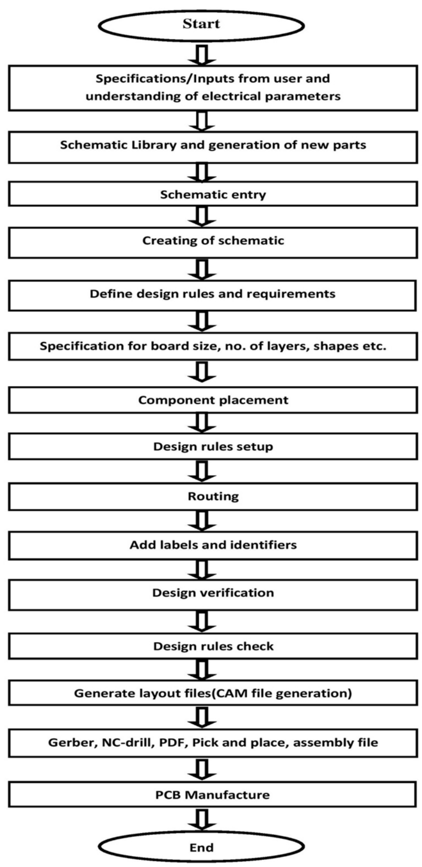


Figure 4: experimental setup

VIII. FLOW CHART



IX. CONCLUSION

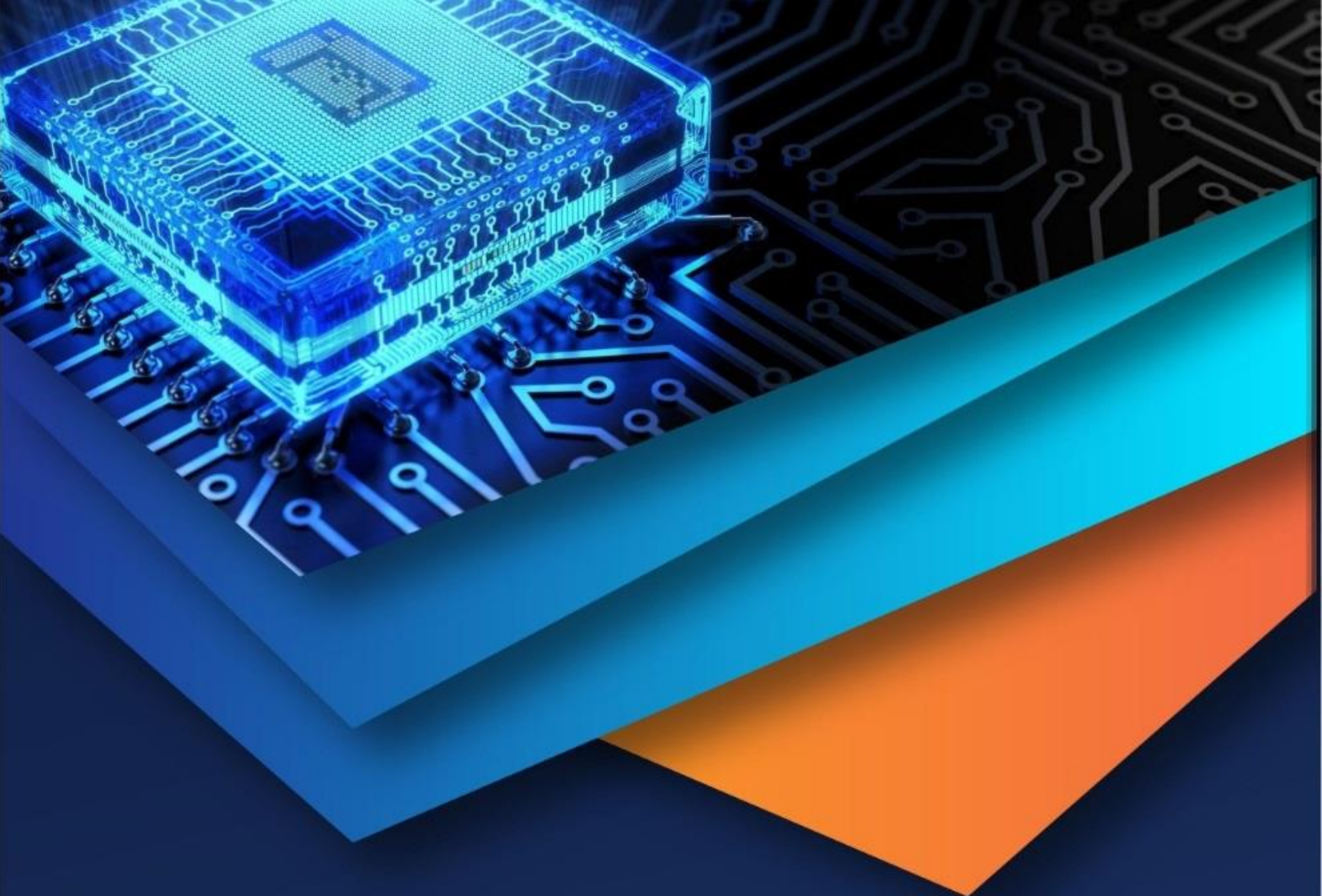
In conclusion, designing a PCB for an Arduino-based system requires a deep understanding of the system's requirements and the principles of circuit design. It is important to choose the right components and layout to ensure proper functionality, reliability, and efficiency.

The design process involves schematic capture, component placement, routing, and testing. It is crucial to consider factors such as power management, signal integrity, and electromagnetic compatibility during the design phase. Adequate documentation and quality checks should be performed to ensure that the final product meets the specifications and regulatory requirements.

By following best practices and considering key factors such as size, power consumption, and functionality, the PCB design for an Arduino-based system can achieve optimal performance and reliability. A well-designed PCB can enhance the system's functionality, reduce development time, and simplify maintenance and upgrades.

REFERENCES

[1] Bates, M.P.,"Programming 8-bit PIC Microcontrollers in C with Interactive Hardware Simulation," NewnessUSA, 2008.
[2] Smith, A.G. "Introduction to Arduino: A Piece of Cake", 2011 Retrieved online 2018 from www.intotoarduino.com
[3] Tutorialpoint. 2016.Retrieved online from www.tutorialpont.com, 2018, 24-35
[4] RAKO Paul, Prototyping Techniques: Things to Know Before Pulling the Trigger, EDN, May 2008.
[5] VERBELEN Yannick, Geïntegreerd Practicum: Robocup Project, unpublished, 2009.
[6] Wolf, W. H. (1994). Hardware-software co-design of embedded systems. Proceedings of the IEEE, 82(7), 967-989.
[7] Zhou, Z., Fu, C., Xue, C. J., Han, S., Zhang, W., & Ju, L. (2023). Optimizing Worst-Case Data Freshness in RF-Powered Networked Embedded Systems. IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems.
[8] ZUMBAHLEN H., Linear Circuit Design Handbook, Analog Devices, ISBN 9-780-750687-03-4, April 2008.
[9] KESTER Walt, High Speed System Applications, Analog Devices, ISBN-10: 1-56619-909-3, ISBN-13: 978-1-56619-909- 4, Part 4, 2006.
[10] MAXFIELD Clive, Altium redefines board-level design, EE Times, February 2011.
[11] Verbelen, Yannick & Belle, Davy & Tiete, Jelmer. (2013). Experimental Analysis of Small Scale PCB Manufacturing Techniques for Fablabs. International Journal of Engineering Innovation and Research. 2. 136-143.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)