|| Volume 9 || Issue 6 || June 2025 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

"A PROTOTYPE OF MARS EXPLORATION AND RESEARCH ROVER"

Mr. SHREYASH DHAWALE[1], Mr. RUSHIKESH JADHAV [2], Mr. DAULAT JAWALE[3], Mr. VEDANT REKHI[4], PROF. Mr. SANDIP GATH[5]
DEPARTMENT OF ELECTRICAL ENGINEERING, RAJIV GANDHI GOLLEGE OF ENGINEERING, KARJULE HARYA, TALUKA PARNER, AHAMADNAGAR[1,2,3,4,5]

---***---

Abstract: The scientific community is very interested in using mini rovers that can travel huge distances and carry multiple science instruments to explore Mars. These rovers would travel to locations that were several kilometres apart, set up equipment against loose rocks or outcrops, look for a sample of interest, and gather soil and rock samples to bring back to Earth. The goal of our study is to create technologies that allow for these kinds of situations while staying within the mission's mass, power, volume, and cost limits. Excellent scientific data on the soil and atmosphere will be provided by stationary landers, yielding knowledge on the planet's climatic history. At most locations, the soil and atmosphere are probably accessible and well-mixed, allowing for effective characterisation without requiring a lot of movement. However, the soil and atmosphere will have significant limitations in terms of comprehending long-term biological and climatic problems. Each one represents a single point on an evolutionary route and is a cumulative record. It is necessary to gather scientific data from the rocks, soil, and atmosphere in the vicinity of the landing site.

We create a MERR (Mars Exploration & Research Rover) prototype as part of our project. The rover uses DC electric motors for its three-wheel driving system and autonomous navigation. Along with sensors like a temperature and distance measurement sensor, it also contains a soil/rock collection arm. For additional study, a roving rover can gather tiny samples of rock and soil from various locations on the exploring planet.

---***---

I.INTRODUCTION:

Aim and Objective

The creation of a Mars exploration and research rover prototype is the goal of this project. It gathers soil from the surface of the planet it is exploring using the soil collecting arm, and it measures the temperature of the atmosphere with the temperature sensor. The temperature and other information about the Rover's operation are shown on an LCD module.

Motivation

The only countries to successfully send spacecraft to the fourth planet from the Sun are the US, Europe, and the Soviet Union. The Mars Orbiter Mission (MANGALYAN), India's interplanetary probe, has departed for the first phase of its scientific exploration journey. Our nation still lacks research and development in the area of planetary explorations. Thus, we made the decision to deploy a research rover prototype.

II.SYSTEM DESIGN

Literature Review

Upon landing on Mars, an automated motor vehicle known as a rover drives itself across the planet's surface. Compared to fixed landers, rovers offer a number of advantages. They can explore a larger area, be guided to intriguing features, position themselves in sunny areas to survive the winter, and increase our understanding of how to operate extremely remote robotic vehicles.

Block Diagram

The block diagram of the prototype of the Mars Exploration & Research Rover is shown in fig. 2.1. It mainly includes

- Drive system
- Power supply and regulator system
- Sensor system
- Main Board (Arduino)
- Error control and feedback system
- LCD Display

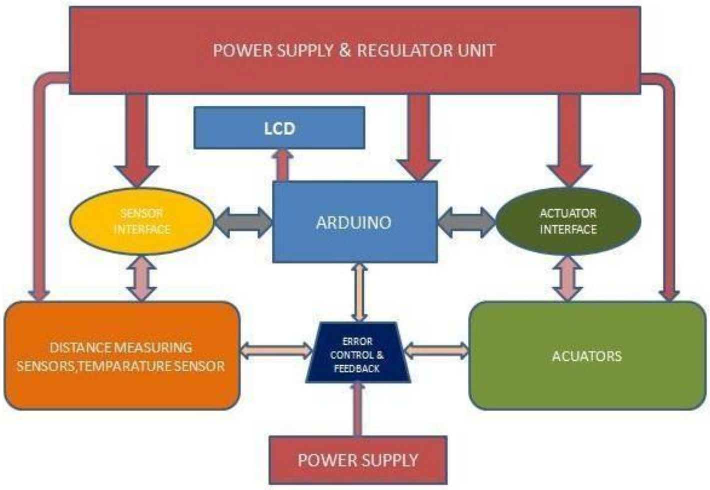


Fig: Block Diagram of Mars Exploration & Research Rover

Rover Programming

The Arduino board, which is the hardware you work with when creating your things, and the Arduino IDE, which is the software you run on your computer, are the two main components of Arduino. An Arduino program, or sketch, is created in the IDE and uploaded to the Arduino board. The board is instructed by the sketch.

Program Description

The Arduino program that is loaded into the microcontroller in our project generates the different control signals. Therefore, every time, the auto navigation algorithm checks all of the sensor outputs. Corresponding loops and control signals are generated in the event that any of the conditions stated in the Auto navigation program are met. The table below lists every control signal produced by the Auto navigation application Hardware Implementation

	Front Sensor	Right Sensor	Left Sensor	Functions
1	High	High	High	Forward
2	Low	High	High	Turn Left
3	High	Low	High	Turn Left
4	High	High	Low	Turn Right
5	Low	Low	High	Turn Left
6	Low	High	Low	Turn Right
7	High	Low	Low	Forward
8	Low	Low	Low	Backward

Table 1 Auto Navigation Program

PCB Development

Using conductive paths, tracks or signal lines etched from copper sheets laminated onto a non-conductive substrate, a printed circuit board, or PCB, mechanically supports and electrically connects electronic components. It is more precisely referred to as printed wire board (PWB) or etched wiring board when the board contains just copper rails and features and no circuit elements such capacitors, resistors or active devices have been produced into the real substrate of the board. Almost all but the most basic commercially made electronic devices today employ printed circuit boards, which also enable completely automated assembly techniques not feasible or viable in past eras tag type circuit manufacturing processes.

PCB Layout

Our project's PCB layout was created with OrCAD on a computer.

Mostly used for electronic design automation, OrCAD is a proprietary tool set. Electronic design professionals and electronic technicians use the program mostly to produce electronic schematics and electronic prints for producing printed circuit boards.

The OrCad Capture CIS tool produced the circuit's schematic layout. OrCAD Capture offers PSpice's quick and easy schematic design entry for analogue simulation or PCB creation. It is connected with the component information system (CIS) to automatically synchronise and evaluate externally acquired part data. The OrCAD Layout Plus program helped to derive the matching PCB layout. Accepting a layout-compatible circuit netlist (such as from Capture CIS), Layout Plus is a circuit board layout tool that creates an output layout file fit for PCB manufacture. *Figure shows a MERR Interfacing System schematic*.

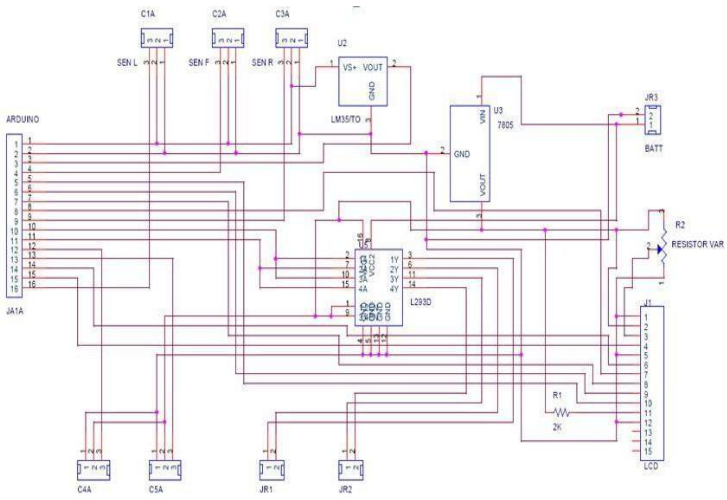


Fig: Schematic Of MERR

PCB Making

1. When Printed Circuit Board (PCB) needs to be produced in limited quantities, their production is as much as technique especially so. PCB layouts and final board drawing can be done in numerous ways. Usually, PCB fabrication calls for two steps.
2. Getting ready for the PCB Illustration
3. Building the PCB straight from the drawing

PCB Drawing

Making the PCB calls for certain basic issues such component arrangement on a piece of paper. Finding holes, determining the diameters of different holes, the best area each component should occupy, the form and position of islands for linking two or more components at a given site, full space uses and prevention of component congestion at a given place. Regarding component anchoring levels, 1mm diameter holes; and for fixing PCB holding screws to the chassis 3mm diameter holes can be made. Following these hints a sketch of the PCB is made.

PCB Fabrication

To prepare a copper-clad PCB, the surface is first cleaned using fine emery or sandpaper. The circuit layout is then traced onto the laminate using carbon paper and fixed with clips. Holes are drilled (1mm or 3mm), and the pattern is coated with black enamel paint. Any paint spills causing shorts are corrected by scraping.

The board is etched by pouring a heated ferric chloride solution (20–22g in 75ml water at ~60°C) over it for about 45 minutes, dissolving unwanted copper. After etching, the paint is removed with thinner or acetone, and the board is washed and dried.

Finally, components are soldered onto the PCB following the wiring diagram—typically, resistors first, followed by ICs.

Soldermask:

Soldermask is a protective layer applied to PCBs during assembly to prevent solder from short-circuiting adjacent tracks. It is usually green, though red and blue are also used. In EDA software, soldermask expansion rules define the spacing between pad edges and the soldermask boundary.

Silkscreen Technology

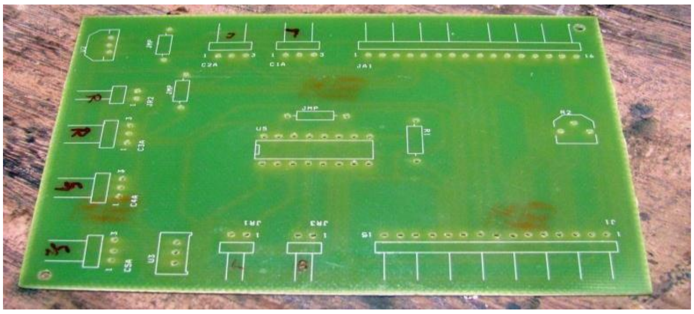
Usually by screen printing epoxy ink in a contrasting colour, line art and text can be printed onto the exterior surfaces of a PCB; alternatively, LPI or dry film can be used, as the solder resist. When space allows, the legend can show component designators, switch setting guidelines, test points, and other elements used in assembling, testing, and circuit board servicing.

Glass Epoxy

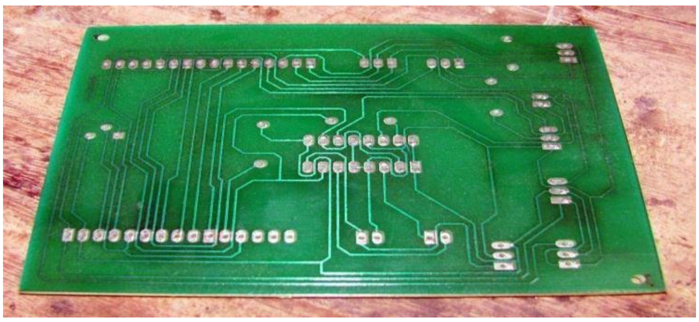
Usually known as FR4 laminate, glass epoxy PCB Glass-reinforced epoxy laminate sheets, tubes, rods and printed circuit boards (PCB) are assigned a grade known as FR-4, or FR4. Comprising woven fibreglass cloth with an epoxy resin binder, FR-4 is a composite material flame resistant—that is, self-extinguishing.Popular and adaptable high-pressure thermoset plastic laminate grade with good strength to weight ratios is FR-4 glass epoxy. FR-4 is most often employed as an electrical insulator having great mechanical strength since its water absorption is almost negligible. In both dry and wet environments, the material is well-known to maintain its great mechanical values and electrical insulating properties. These qualities, together with superior manufacturing quality, help this grade to be useful for a broad range of mechanical and electrical uses.

Usually using bromine, a halogen, FR-4 epoxy resin systems help FR-4 glass epoxy laminates to have flame-resistant characteristics. G-10 non flame resistant nevertheless finds use in some applications where thermal degradation of the material is a desired feature.

Final PCB: The final PCB of interfacing part of our rover is shown in figure given below. Figure (a) shows the component side of the PCB and figure(b) shows the soldering side of PCB



Fig(a):Component Side of PCB



Fig(b) Soldering Side of PCB

Soldering

Soldering is a process in which two or more metal items are joined together by melting and flowing a filler metal (solder) into the joint, the filler metal having a lower melting point than the workpiece. Soldering differs from welding in that soldering does not involve melting the work pieces. In brazing, the filler metal melts at a higher temperature, but the workpiece metal does not melt. Formerly nearly all solders contained lead, but environmental concerns have increasingly dictated use of lead-free alloys for electronics and plumbing purposes.

Requirements of Soldering

Soldering Iron

- Soldering iron is the heat source used to melt solder. Irons of the 15W to
- 30W range are good for most electronics/printed circuit board work..

Solder

Solder is a fusible metal alloy used to join together metal workpieces and having a melting point below that of the work piece(s).

Flux: The purpose of flux is to facilitate the soldering process..

Preparing To Solder:

Tinning The Soldering Tip

Before use, a new soldering tip, or one that is very dirty, must be tinned. "Tinning" is the process of coating a soldering tip with a thin coat of solder. This aids in heat transfer between the tip and the component you are soldering, and also gives the solder a base from which to flow from.

Step 1: Warm Up The Iron

Warm the soldering iron or gun thoroughly. Make sure that it has fully come to temperature because you are about to melt a lot of solder on it. This is especially important if the iron is new because it may have been packed with some kind of coating to prevent corrosion.

Step 2: Prepare A Little Space

While the soldering iron is warming up, prepare a little space to work. Moisten a little sponge and place it in the base of your soldering iron stand or in a dish close by. Lay down a piece of cardboard in case you drip solder and make sure you have room to work comfortably.

Step 3: Thoroughly Coat the Tip in Solder

Thoroughly coat the soldering tip in solder. It is very important to cover the entire tip. You will use a considerable amount of solder during this process and it will drip, so be ready. If you leave any part of the tip uncovered it will tend to collect flux residue and will not conduct heat very well, so run the solder up and down the tip and completely around it to totally cover it in molten solder.

Step 4: Clean the Soldering Tip

|| Volume 9 || Issue 6 || June 2025 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

After you are certain that the tip is totally coated in solder, wipe the tip off on the wet sponge to remove all the flux residue. Do this immediately so there is no time for the flux to dry out and solidify.

How to Solder

Soldering A Printed Circuit Board (PCB)

Soldering a PCB is probably the most common soldering task an electronics hobbyist will perform. The basic techniques are fairly easy to grasp but it is a skill that will take a little practice to master. The best way to practice is to buy a simple electronics kit or assemble a simple circuit (such as an LED chaser) on a perf-board. Don't buy that expensive kit or dive into a huge project after only soldering a few joints.

Soldering components onto a PCB involves preparing the surface, placing the components, and then soldering the joint.

- Step 1: Surface Preparation:
- Step 2: Component Placement
- Step 3: Apply Heat
- Step 4: Apply Solder to the Joint
- Step 5: Inspect The Joint and Cleanup

Tips and Tricks

These tips should help you become successful and get down to some serious building.

1.Use heatsinks. Heatsinks are a must for the leads of sensitive components such as ICs and transistors. If you don't have a clip on heatsink, then a pair of pliers is a good substitute.

2.Keep the iron tip clean. A clean iron tip means better heat conduction and a better joint. Use a wet sponge to clean the tip between joints.

3.Double check joints. It is a good idea to check all solder joints with an ohm meter after they are cooled. If the joint measures any more than a few tenths of an ohm, then it may be a good idea to resolder it.

4.Use the proper iron. Remember that bigger joints will take longer to heat up with an 30W iron than with a 150W iron. While 30W is good for printed circuit boards and the like, higher wattages are great when soldering to a heavy metal chassis.

5.Solder small parts first. Solder resistors, jumper leads, diodes and any other small parts before you solder larger parts like capacitors and transistors. This makes assembly much easier.

Rover Body Design

The rover platform is constructed using sheet metal. The width of the platform is 21 cm and the length is 42 cm. The thickness of the sheet metal used is 0.08 cm. The PCB, Arduino Board, steering control servo, soil collecting arm servo, battery pack, drive system motors and the distance measuring sensors are attached to this sheet metal platform. The upper portion used for covering the rover is constructed using black acrylic plastic of length 36 cm, breadth 19cm and height 6cm. The LCD module is attached at the back top portion of the covering in an inclined manner using a special design.

System Integration

The different components of the rover such as distance measuring sensors, steering wheel, soil collecting arm, drive wheels, and battery pack are interfaced with the PCB. The solar panel and the LCD module are also integrated in the metallic body.

Final Rover

The figure (a)shows the disassembled view of Rover and figure (b) shows the assembled view of Rover.

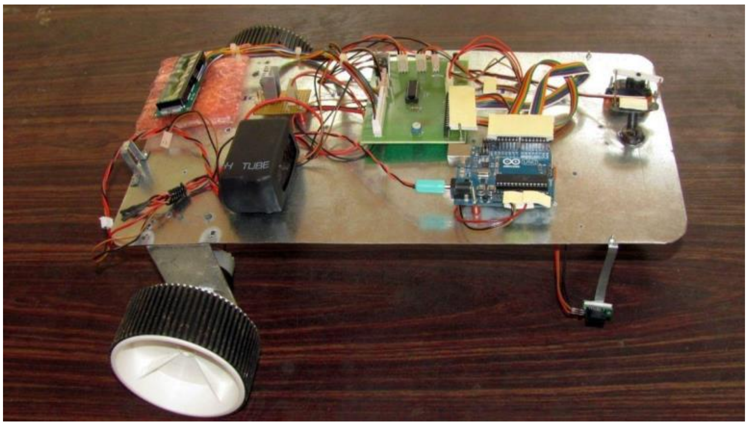


Fig (a) Disassembled view of MERR

Fig (b) Assembled view of MERR

III.CONCLUSION

Thus the project "A Prototype Of Mars Exploration And Research Rover" has been completed successfully. The drive system, power supply and regulator system, sensor and actuator system, main board Arduino and the LCD module were analyzed and studied. The microcontroller was programmed with the autonavigation control program for the proper activities of MERR. As any type of transmission or reception devices are not used. So the LCD module was used to display the activities of MERR.

IV.REFERENCES

[1] Microrover Research for Exploration of MARS, Samad Hayati, Richard Volpe, Paul Backes, J. Balaram, and Richard Welch, Jet Propulsion Laboratory , California Institute of Technology.

[2] Autonomy for Mars Rovers: Past, Present, and Future, Max Bajracharya, Mark W. Maimone, and Daniel

Helmick, Jet Propulsion Laboratory, California Institute of Technology

[3] Wikipedia

[4] Getting Started With Arduino, Massimo Banzi

[5] Beginning Arduino, Michael McRoberts

[6] Arduino Cookbook, 2nd Edition, Michael Margolis