Redesigning Metal Motor Control Panel with Plastic

Varad Kulkarni[1] Sahil Kesanakurty[2] Dr. Dattatraya Bhise[3]
[1,2,3]Department of Mechanical Engineering
[1,2,3]JSPM Narhe Technical Campus, Pune, India

Abstract — This project encompasses many aspects and has several parts. We are working on an industrial project which will be implemented by the company- "Tech Arch Pvt. Ltd." located in Chakan. Primarily, the project focuses on introducing a new product into the market as a replacement to an already existing product (a panel which holds the controller, electrical parts and switches which operate a motor remotely). This existing product has a few shortcomings which have been overcome in our new product. Also, a lot of emphasis is given to reducing the cost of the product by implementing several methods like easing the manufacturing flow, choosing proper material and design, integrating two product designs to have a common mould for both, reducing child parts and removing extra steps in manufacturing. Our motive to pursue this project was to utilise prior knowledge in plastic injection moulding gained from internships in the field and recognise and fulfil a scope for a successful product which will prove profitable. The result will be a safe, affordable, lighter and aesthetic product which will fulfil all needs of the customers.

Keywords: Redesigning Metal Motor, Control Panel, Plastic

I. INTRODUCTION

The market for manufacturing products primarily suitable to the agricultural field is very competitive; every new product of the companies undercuts the previous Current Metal Motor Panel by a margin because the agricultural products market is very profitable. We are delving into the market by undercutting previous products by studying about the scopes of improvement, studying the customer requirement and whether it is being met by the Current Metal Motor Panel or not, making the design, and finally delivering it satisfying customer expectations. Existing motor control panel is made of metal and is used to control the water pump. Currently there are several similar products of similar dimensions having very minor design changes, we recognised that there is also a scope for integrating the designs of 2 products into a single mould, so the mould can be interchangeable, thus saving costs of having a separate mould.

Another area of improvement found was that there are many parts in the product where time was being wasted since it required constant and repetitive alignment of child parts into the body, in the long run costs time to the company and thus revenue loss, we propose to overcome these steps by making suitable changes to the design and implementing new ideas to reduce manufacturing and assembly steps. Many problems were noted such as electric shocks, rusting and excessive product weight. The major application of the product is near a water body so it is constantly exposed to the elements like direct sun, heavy rainfall and constant exposure to moisture droplets, metal being a good conductor of electricity and having high moisture in the air causes current to pass through the metal due to which several accidents happened by causing incidents like short-circuit and damage of control panel and also rusting.

In subsequent chapters, we will discuss all the steps to avoid such problems while keeping the product affordable and safe. So this project aims towards designing a better, safe, implementing various technologies, and providing a feasible alternative to the metal control panels.

A. Problem Statement:

1) Redesigning Metal Motor Control with Plastic:

"High cost, safety hazard, less life, low weight to strength ratio, rusting, excessive and lengthy manufacturing process, excessive time of production per part are some of the problems in the Current Metal Motor Panel, all these shortcomings are fulfilled in our product"

B. Existing Metal Motor Control Panel:

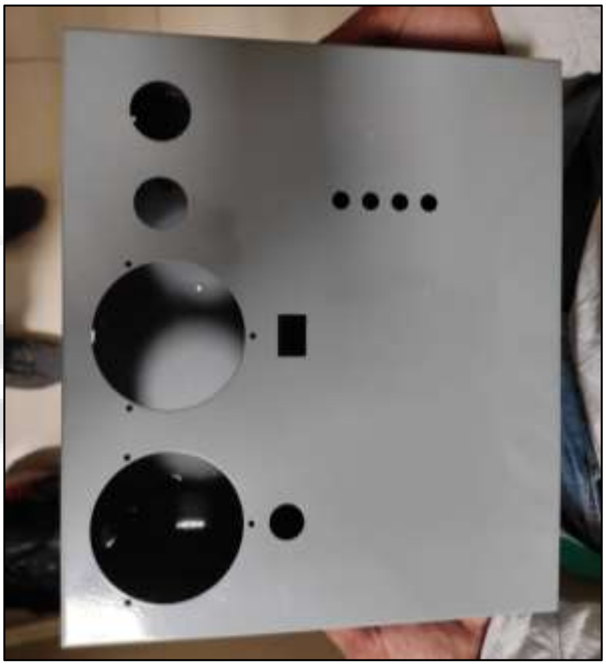

Fig. 1.1: Top View- Closed Metal Motor Panel

Fig. 1.3: Current Metal Motor Panel in market (Motor control panel)

1) Problems on Customer's side:

- Product finds its application mostly in farmlands and near a water body like wells, so the body starts to rust with time and exposure to moist air.
- Continuous customer complaints regarding electrical shocks since existing material is metal.
- There is always a demand and inclination of customers towards a cheaper alternative which also not only provides the same but better quality.

2) Problems on Manufacturer's side:

- Since plastic injection moulding is planned, so instead of using separate moulds for separate products - integrating production of two types of control panel using single mould is planned, thus saving cost and manufacturing time simultaneously.
- High cycle time of producing every part due to need of time consuming steps like alignments of child parts.
- Additional expenses since there will be many child parts

C. Objective:

To deliver a product which satisfies all requirements of customers at the same time being affordable than the competition:

- To carry out literature review of injection moulding.
- To find out the best suitable material to replace current material.
- Design the Motor control panel using CAD software and analysis using ANSYS software using suitable dimensions.
- Design mould for manufacturing the designed control panel.
- Increase the value of our product in terms of affordability and safety.
- Making sure that our design is being manufactured at all required tolerances by the tool maker by testing.

D. Scope:

At the end of the stipulated time of the project's duration - the product delivered will be:

- Plastic injection moulded
- Aesthetic
- Having safe design theoretically and physically
- Cheaper than the Current Metal Motor Panel

E. Considerations and Tasks to be undertaken:

- Finalise the most suitable and safe design and dimensions through trial and error
- Make sure there is timely deliverance of the design to the tool maker.
- Make sure the moulds are being manufactured according to design.
- Undergo testing of the moulds

F. Methodology:

- Studying Current Metal Motor Panels and identifying the problems in it.
- Noting down the problems faced by customers.
- Studying the complete manufacturing flow of products.
- Studying and researching the scope for overcoming the problems.
- Literature study.
- Consider various study materials.
- To study the properties of considered material.
- To find the best material suitable for the product.
- Started designing motor control panel according to dimensions and carried out analysis on the same.

G. Design Considerations for Plastics:

1) Drafts

Draft is the amount of taper on the vertical walls of the plastic part. Without a draft, a part may not eject from the mould or may sustain damage during ejection. Typically, draft angles between 1° and 2° are required but can vary depending on part restrictions and specifications.

2) Ribs

A plastic part that has been designed with a minimal wall thickness will not be as strong as a thicker part – which is why the inclusion of ribs may be needed to help reinforce the part's strength. Depending on the material used, rib thickness should be between 50 – 70 per cent of the relative part thickness to avoid sink marks. To avoid sinking, designers may core out material to reduce defect risk.

3) Bosses

A boss feature finds use in many part designs as a point of attachment and assembly. The most common variety consists of cylindrical projections with holes designed to receive screws, threaded inserts, or other types of fastening hardware.

4) Fillet

The fillet radius should be 25-60% of the nominal wall thickness. The outside corner radius should equal the inside radii plus the wall thickness. Helps keep uniform wall thickness and avoid stress concentrations.

5) Wall Thickness

Designing your part so that wall thickness is consistent can help avoid many part defects that can occur during the manufacturing process. When the plastic melts, it flows to the areas of least resistance. If your part has inconsistent thicknesses throughout, the melt may flow into the thick areas first (depending on gate locations). When this occurs, the thin areas may not fill properly. Additionally, thicker areas tend to cool more slowly and are at risk for voids or sinking defects. Designing your part with rounded corners will also aid in the proper filling of the part during the moulding process.

6) Mould shrinkage

The shrinkage that occurs during the plastic part moulding process can be as much as 20 per cent by volume. Crystalline and semi-crystalline materials are most prone to thermal shrinkage.

7) Gate Location

- Gate thickness should be 50 –80% of the wall thickness. Gate thickness controls packing time.
- Gate width should be twice the gate thickness.
- Gate lands should be kept to a minimum.
- Gates should be located at right angles to the runner to allow the polymer melt to impinge against the mould wall.

H. Material Research:

The injection moulding process offers a vast array of plastic resins to work with. These materials have been developed to meet specific requirements for strength, high temperature and chemical resistance, abrasion resistance and low friction. Thermoplastics resins fall into two distinct categories, each with its advantages and disadvantages.

1) Amorphous plastics: for clear and opaque parts and include polycarbonate, acrylic, PETG, ABS and polysulfone
   – Advantages – Bonds well-using adhesives, very stable dimensionally and good impact resistance.
   – Disadvantages – Low resistance to fatigue and stress cracking.
2) Semi-crystalline thermoplastics: These include the polyethene family (LDPE, HDPE, and UHMW-PE), Polypropylene, nylon, acetal and fluoropolymers.

– Advantages – Excellent for bearing, wear and structural applications, good chemical and electrical resistance, and lower coefficient of friction.
– Disadvantages – Difficult to bond with adhesives, only average impact resistance.

These resins can be categorized as a) low-cost commodity resins, b) medium-cost engineering resins and c) high-cost high-performance/speciality resins (see table below).

AMORPHOUS RESINS	SEMICRYSTALLINE RESINS	COST
High-Performance/Speciality Polyetherimide or PEI Strength: High Heat & Chemical Resistance: High Applications: Aerospace	High-Performance/Speciality Polyetheretherketone or PEEK Strength: High Heat & Chemical Resistance: High Applications: Bearings, medical implants	Expensive
Engineering Polycarbonate or PC Transparent Strength: Moderate Heat Resistance: High Electric Insulator: High Applications: Electrical, windows	Engineering Polymide or PA (Nylon) Strength: Moderate to high Chemical Resistance: High Abrasion Resistance: High Shrinkage and Warp: Low Applications: Auto parts, textiles	Moderate
Commodity Polystyrene or PS Transparent Strength: Low Heat Resistance: Low Application: Cutlery, cups	Commodity Polypropylene or PP Flexibility & Toughness: High Chemical Resistance: High Fatigue Resistance: High Applications: Bottles, crates & cases, living hinges	Inexpensive

Table 1.1: Material Specifications

1) Why choose PC-ABS:

– High impact strength even at low temperatures
– Heat resistance
– High stiffness
– Easy processing
– Low overall shrinkage and high dimensional accuracy
– PC-ABS is the most cost-effective alternative to PC
– This blend is 5-60% stronger than ABS
– This makes it ideal for producing durable parts for projects like industrial equipment and manufacturing
– Thus we have chosen PC-ABS as the type of plastic.

Fig. 1.4: PC-ABS Materials

I. Research into ways of creating the PC-ABS Stronger:

1) Fillers & Additives:

Part strength can be improved by the addition of reinforcing materials such as glass fibres. Fillers such as glass fibres can be added to many resins to make them stronger and stiffer, while at the same time making them slightly more brittle. Other fillers such as talc can be used to increase the hardness of moulded parts. Other fillers such as glass beads and fused silica can be used to reduce the flexibility of a part, reduce warpage or minimize shrinkage. Additives such as UV stabilizers, static dissipating agents, or flame retardants can be added to the resin compound if required.

Fig. 1.5: PC-ABS material with additives

J. Concept application:

We have applied with the concept of "Compliant Mechanisms" to our project.

1) Compliant Mechanisms:

It is always thought that rigidity is necessary for any good quality parts, but compliant mechanisms disagree with this. Compliant mechanisms use the flexibility of parts to their advantage in other words it is "A mechanism is a mechanical device used to transfer or transform motion, force, or energy".

Traditional rigid-body mechanisms consist of rigid links connected at movable joints, this increases complexity and cost, but compliant mechanisms overcome these and many other disadvantages which traditionally machined parts/mechanisms have.

Traditional mechanisms gain mobility with the help of movable joints. Compliant mechanisms, however, achieve the same result from the deflection of flexible members. Like rigid-link mechanisms, they are also devised to transfer or transform motion, force, or energy, but they do not rely on movable joints, hinges, or pins to do so. The difference is in the design, in which flexible members do the same job. A Compliant System need not be flexible at all links, but some important ones must be flexible. In a compliant mechanism, transmission and/or transformation of motion and force is achieved through elastic deformation of the members of the mechanism.

Fig. 4.1: Complaint Mechanism

2) Our Implementation:

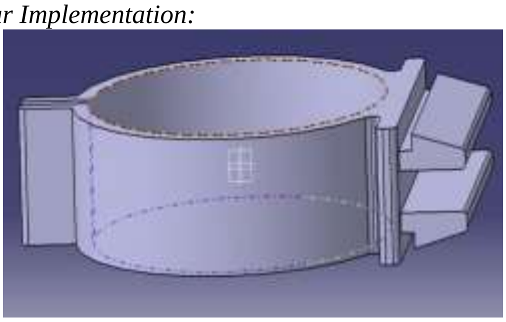

Fig. 4.2: Proposed complaint mechanism design

The Jaws at the right side are designed such that they bend when loaded and the direction of bending is such that it bends inwards and locks onto the body of the casing.

– Advantages:
  - Reduces child parts like screws.
  - Reduces excess time wasted in aligning screws
  - Improves efficiency of production
  - Reduces cost incurred
– Disadvantages:
  - Since plastic is used, it is weak
  - Complex design
  - Less efficiency compared to metal

II. LITERATURE REVIEW:

A. Injection Moulding:

1) Injection Moulding Machine:

The injection machines are machines that melt plasticize the moulding material inside the heating cylinder and inject this into the mould tool to create the moulded product by solidifying it inside it. The injection machine is constructed of a mould clamping device that opens and closes the mould tool, and a device that plasticizes and injects the moulding material. There are several types of injection machines, and the difference is made by how these two devices are arranged.

– Horizontal injection machine: Both mould clamping device and injection device compounded horizontally
– Vertical injection machine: Both mould clamping device and injection device compounded vertically
– Two-colour injection machine
– Rotary injection machine
– Low foam injection machine
– Multi material injection machine
– Sandwich injection machine

2) Selection of injection moulding machine:

– Select by injection volume

As a guide, generally the injection machine should be selected so that moulded product volume will become 30% to 80% of the machine's injection volume. When moulding, the relation of the machine's injection volume Q (g) and one shot weight (sprue and runner weight included) W (g) should be in the range indicated below.

$$Q = (1.3 \sim 1.5) \times W$$

If the injection volume is too small, plasticization will not make it, and might lose its original physicality as a moulded product because the resin will be sent without enough plasticization. On the other hand, if the injection volume is too big, residence time inside the cylinder will be longer and cause degradation by more chance.

– Select by mould clamping pressure

Both toggle type and direct pressure type is suitable when moulding NOVADURAN. The relation of moulded product projected area a (cm2) and required mould clamping pressure P (ton) should be in the range indicated below.

$$P = (0.5 \sim 0.7) \times A$$

B. Injection Moulding Process:

Rushikesh D. Dhomne, Abhijeet A. Raut, Kalyan Labade [2]

1) Injection Moulding Process:

This is a manufacturing process for produced plastic product from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mould cavity by a reciprocating screw or a ram injector, where it cools and hardens to the configuration of the mould cavity. The mould is usually constructed using either steel or aluminium, and precision-machined to form the features of the desired part. This process is widely used for manufacture the variety of parts, from the smallest product to entire body panels of cars. The most common method of production, with some commonly made items including computer components to outdoor furniture. These machines have many components and are available in different configurations, including a horizontal configuration and a vertical configuration. However, regardless of their design, all injection moulding machines utilize a power source, injection unit, mould assembly, and clamping unit to perform the four stages of the process cycle. Injection moulding machines consist of a material hopper, an injection ram or screw-type plunger, and a heating unit. They are also known as presses, they hold the moulds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can exert. This force keeps the mould closed during the injection moulding process. Tonnage can vary from less than 5 tons to over the 9,000 tons, with the higher Fig. used in comparatively few manufacturing operations. The total clamp force needed to the determined by the projected area of the part being moulded.

C. The Mould Quoting Process:

The quoting process for plastic parts can be difficult for both the mould customer and supplier. Consider the view of the mould customer. The procurement specialist for the product development team sends out requests for quotes (RFQs) to several mould makers. After waiting days or weeks, the quotes come back and the customer discovers that the development time and cost of the mould may vary by a factor of 3 or more. In such a case, prospective mould purchasers should ask about the details of the provided quotes, and check if the costs can be reduced through product redesign. To reduce uncertainty related to pricing and capability, many prospective customers maintain a list of qualified suppliers, who tend to provide faster turn-around, more uniform quality, and better pricing across multiple projects. Long-term, trusting partnerships can provide for rapid application and mould development by avoiding the quoting process altogether and invoicing on a labour cost plus materials cost (referred to as “cost plus”) basis.

Now consider the view of the mould supplier. The mould designer must invest significant time developing a quote that may have a relatively small chance of being accepted. Sometimes, the mould designer may have to redesign the product and perform extensive analysis to provide the quote. While the quote may seem high to the prospective customer, the design may correspond to a mould of higher quality materials and workmanship that can provide a higher production rate and longer working life than some other lower cost mould. If the supplier is extremely busy or idle, then the estimated number of hours and/or hourly rate may be adjusted to either entice or to discourage the potential customer from accepting the quote. Such adjustments should be avoided since the provided quote does not represent the true costs of the supplier, which would become the basis in a long term and mutually beneficial partnership between the mould supplier and the customer. The provided quote typically provides payment and delivery terms for the mould(s) and perhaps even the moulded part(s). A typical mould purchase agreement may specify that the cost of a mould is paid in three instalments:

- The first third: on acceptance of the quote (after which the mould base and key materials are typically purchased);
- The second third: half-way through the mould making project (often when cavity inserts have been machined); and
- The final third: upon acceptance of the quality of the moulded parts.

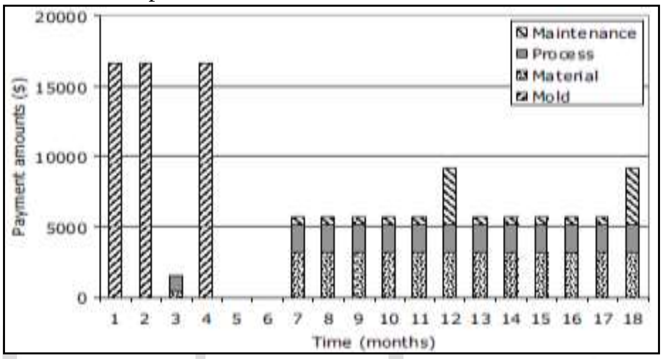

Fig. 2.1: Schedule of mould and moulding expenses

After the mould is purchased, moulds are typically shipped to the specified moulder or the customer’s facility where the parts are moulded and marginal costs are incurred on a per part basis. The cash outlays for a typical project are plotted in Fig. 3.1 on a monthly basis. The material and processing costs in month 3 are related to moulding trials to validate and improve the mould design; a hundred or so pre-production parts may be sampled at this time for marketing and testing purposes. Later, monthly costs are incurred related to production. Maintenance costs may appear intermittently throughout production to maintain the quality of the mould and mouldings. There has been a trend in the industry towards large, vertically integrated moulders with tightly integrated supply chains who can supply moulded parts (and even complete product assemblies). As such, the structure of the quote can vary substantially with the structure of the business. With a vertically integrated supplier, there is typically an up-front fee for the costs associated with the development of the mould, followed by a fee for each moulded part. To protect the supplier, contracts are typically developed that specify minimum production quantities with discounts and/or fees related to changes in the production schedule. Since the structure and magnitude of quotes will vary substantially by supplier(s), a prospective buyer of plastic parts should solicit quotes from multiple vendors and select the quote from the supplier that provides the most preferable combination of moulded part quality, payment terms, delivery terms, and service.

D. Mould Designing:

Designing of moulded products should be done to fulfil demanded characteristics of desired product, and need to evaluate material's practical physicality, mouldability, liquidity, and mould designing condition, comprehensively. Basic points of mould designing are indicated below.

1) Try not to make the thickness excessively thick, and try to keep it even, so that rapid change in thickness will not occur.

If the moulded product thickness is too thick, it will be the cause of defect like sinks marks and void. Also, it will take time to cool down and the moulding cycle will be longer. When there is need to be thick for function, try to keep it even by placing the recess. If there is unevenness or rapid change in thickness, flow marks might occur because it blocks the resin flow, or warpage might occur by uneven mould shrinkage ratio, or deformation of moulded product might occur by uneven cooling speed.

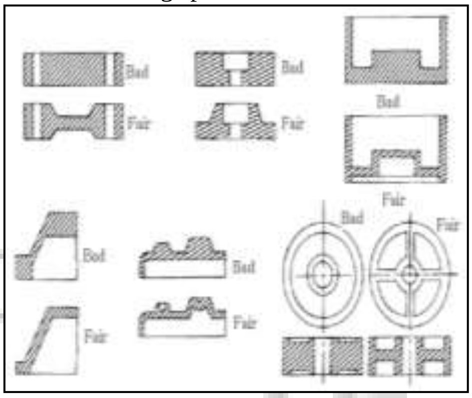

Fig. 2.2: Design of thickness distribution

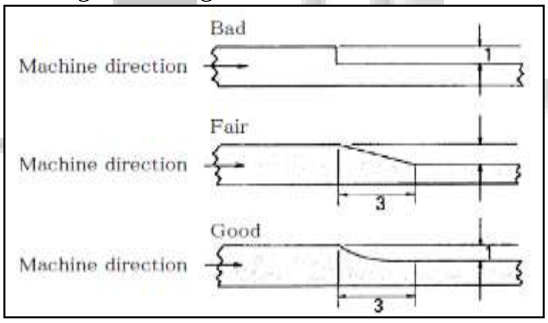

Fig. 2.3: Design of thickness change

2) Try not to make the undercut.

If there is undercut in a moulded product, problems are likely to occur when releasing the mould, so as a general rule, there should be no undercut. When undercut has to be placed by necessity, make the undercut volume small enough towards the limit strain based on material physicality, or design the mould construction not be forced extraction by placing the slide core.

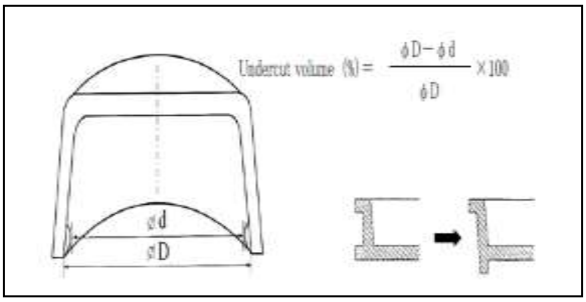

Fig. 2.4: Undercut volume

3) Consider the draft angle.

If the draft angle is not enough, resist when releasing the mould will be big and the moulded product might deform by the ejector pin, so the draft angle should be taken as big as possible. Typical draft angle of NOVADURAN is about 0.25 to 1°in unreinforced grade, and about 0.5 to 2°in GF reinforced grade. Following shows the points of the draft angle design in typical form:

- Box top foam
- Boss
- Grid
- Rib
- Texture

4) Consider the weld location and the resin orientation, and determine the gate location, direction, and number.

5) Try to make the moulded product form tough

6) Consider the easiness of mould creation, and be careful of the following points when forming so that mould machining and finishing will be easier.

- Parting line should be a simple straight line without a tilt.
- Round every edge, both inside and outside.
- If mould machining is difficult because of the shape, use embedded structure.
- Select a shape such as straight line and circle, so that deciding by machining will be simple.
- Decide surface finishing by parts, to get rid of unnecessary face accuracy.

E. Ejector Pin Location and Design:

Without ejector pins, it is usually not possible to remove the moulded part from the mould. The placement of the ejector pins is almost as critical as the location of the gate. The pins should push the part out of the mould without distorting it and without leaving an objectionable mark on the part. A secondary reason for having ejector pins is to aid in the venting of the mould. Ejector pins should be located in the deepest points of the cavity or core. We specifically suggest that ejector pins be located on the deepest points of ribs and bosses. If ejector pins are not located correctly, the part has to be “pulled” out of the deeper areas or the mould. Parts that have to be “pulled” out of the mould are more likely to stick or be distorted during ejection.

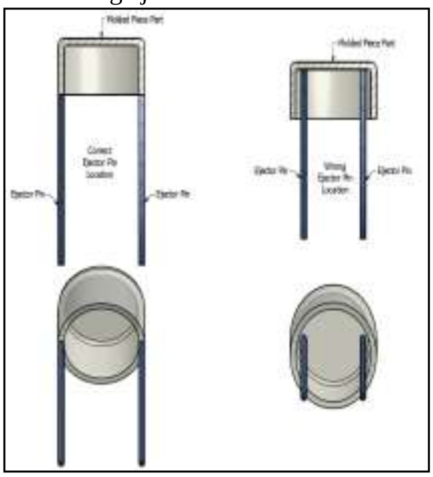

Fig. 2.6: Ejector Pin

Mr. P. Vinod, Mr. K. Vijaykumar has designed a multi cavity injection mould with HRS and CRS. They studied the effect of runner systems, mould cooling and venting by comparison of both the designs. Moulding analysis is carried out using ANSYS. [3]

G.Rajendra Prasad, Dr. S. Chakradhar Goud has studied the dynamic characteristics such as the pressure of the hot runner nozzle using FEA analysis. The basic model was used to compare with the updated finite element model representing the real structure. The FE model presented an average of 40% control in damage of hot runner nozzles than the real cold runner nozzle.[3]

N. Divya, Dr. V. S. Parameswara Rao, Dr. S. N. MalleswaraRao concluded in their paper as the hot runner system accommodates the molten plastic material coming from the barrel and guides it into the mould cavity. Its configuration, dimensions and connection with the moulded part affect the mould filling process and, therefore, largely the quality of the product. The runner system dictates part quality and productivity. In the present work, structural and thermal analysis carried out on original and modified designs of the mould and the results reveals that the deformation, stress, strain and thermal deformations are improved and they conclude that the modified design gives the best output.

Rashi A. Yadav, S. V. Joshi, N. K. Kamble has done the review of the recent research in designing and determining process parameters of injection moulding. A number of research works based on various approaches have been performed in the domain of the parameter setting for injection moulding. They have determined optimal process parameter settings that critically influence productivity, quality, and cost of production in the plastic injection moulding (PIM) industry.

P.K. Bharti, M. I. Khan, presented a review of research in the determination of the process parameters for injection moulding. They done research works based on various approaches including mathematical model, Taguchi technique, Artificial Neural Networks (ANN), Fuzzy logic, Case Based Reasoning (CBR), Genetic Algorithms (GA), Finite Element Method (FEM), Non Linear Modelling, Response Surface Methodology, Linear Regression Analysis.

A. Demirer, Y. Soydan have investigated the effects of the hot runner system on the injection moulding process and properties of the injected part from various aspects in comparison with conventional runner systems. It was observed that the required injection pressure was considerably lower to produce samples with higher weight in the case of the hot runner system. The shrinkage and warpage increased with increasing process temperature, decreased with increasing injection pressure, and occurred at a low level when the sample weight was high. [7]

Gurjeet Singh, Ajay Verma have studied primary processing conditions of moulding from concept development to manufacturing of the product. They studied the effect of various factors based on processing parameters. Quality and productivity are the two important contradictory objectives in any machining process. Some extent of quality has to be compromised while assurance is given for high productivity. It is concluded that productivity decreases while the efforts are channelized to enhance quality. Machining parameters need to be optimized to ensure high quality and productivity. Authors have studied various responses to the quality of the injection moulding process based on performance parameters and methods.

Mehdi Moayyedian, Kazem Abhary have introduced a new geometry of the gate system in the injection moulding process. It has been observed that the corners of existing edge gates result in the turbulence of molten plastic into the cavity which leads to internal and external defects. The new geometry has been introduced to reduce the internal and external defects of injected parts. The target of the study was to have easier de-gating of the final part for the new geometry which decreases the visible blemish of the final part after de-gating. Author suggested a new design of edge gate is suggested for two circular flat plates with a thickness of 1mm. The contribution of this research was to modify the geometry of existing edge gate by removing the corners of the rectangular edge gate to reduce the scrap rate of injected parts. It was observed that also, a smooth flow of molten plastic into the cavities reduces the internal and external defects. The outcome demonstrates filling the cavities with no short shot. Also, no weld lines, meld lines or sink marks were detected with the new design of the edge gate. Finally, the experimentation results in an injected part with a new design of the edge gate which has less visible blemish in comparison with existing edge gate.

F. Thermoset Injection Mould Design Tips:

When designing a mould for an injection moulded part, it is important to keep in mind that the goal is to produce parts with the best quality, at the best cycle possible, with minimum scrap. To achieve this goal, you will need a mould that has a uniform mould temperature, balanced fill, is properly vented and is designed to dimensionally compensate for material shrinkage in all axes.

1) Mould Heating:

A uniform mould temperature means that the temperature of each half of the mould is within +/- 5°F (3°C) for all locations when the mould is heated by oil or steam. Moulds that are heated with electric cartridge heaters can vary by as much as 10°F (6°C). A mould with a uniform temperature will fill more easily and produce parts with less warpage, improved dimensional stability and a uniform surface appearance. Achieving a uniform mould temperature is dependent on the method of mould heating.

A mould that is correctly designed and heated by steam or oil will have a uniform mould temperature because the heat source maintains a constant temperature. However, oil as a heat source is only about half as efficient as steam. Therefore, when using oil to heat a mould, it is necessary to set the oil temperature higher than the desired mould temperature.

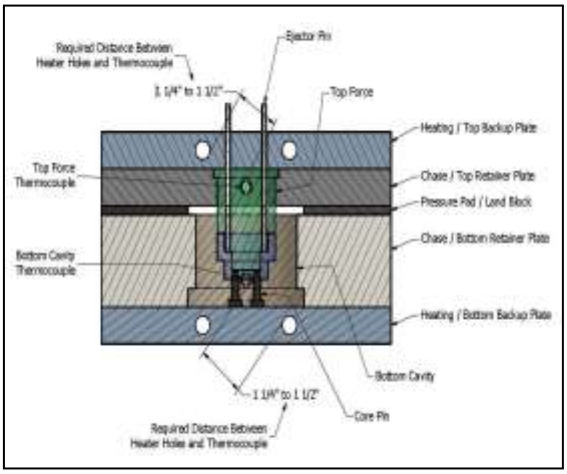

Fig. 4.1: Mould Design

III. PRODUCT DESIGN:

A. *Body:*

- Upper Lid and Lower Lid:

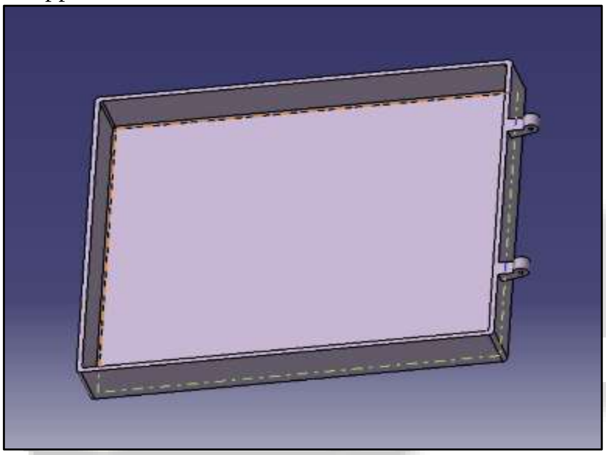
Fig. 3.1: Upper lid

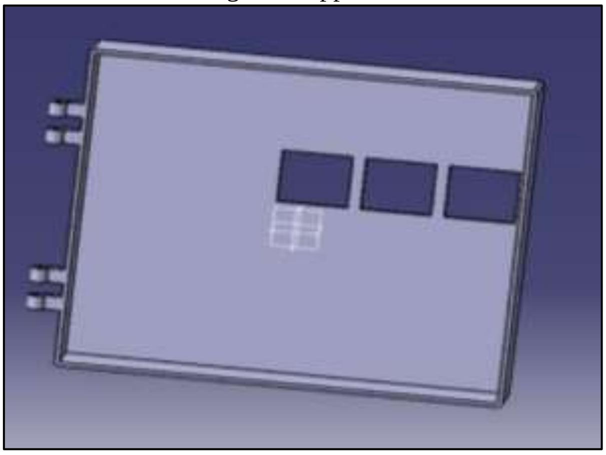
Fig. 3.2: Lower lid

This is the upper lid of the Motor Control Panel designed using CAD software by considering the dimensions given by the Industry. Similar to the Upper Lid, the Lower lid was also designed using CAD Software with two hinges to hold the support the Upper lid

B. *Clamp:*

1) *Iteration No.1:*

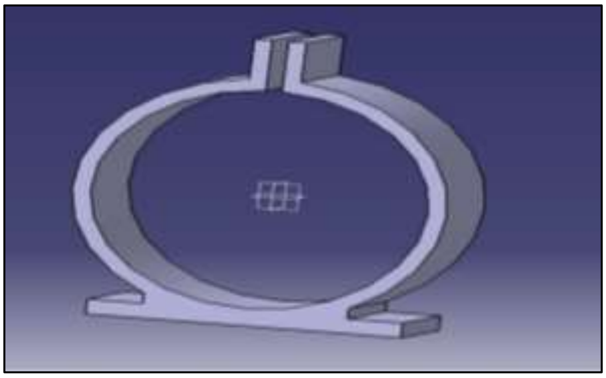
Fig. 3.3: Clamp for holding the components

The first iteration of the clamp which is used to hold the component which is inside the control panel.

(In this design, the Holder was supposed to hang in the panel slots as shown in the Fig. 3.2. This design had less complications unlike a snap-fit design since the manufacturing is easy, but the Panel remains open and internal components are exposed to atmosphere which goes against the Client requirements.)

2) *Iteration No. 2:*

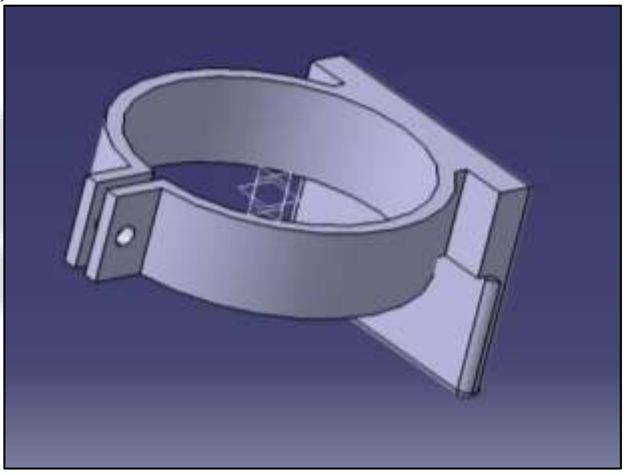
Fig. 3.4 Iteration No. 2 of Clamp for holding the components

Iteration 2 in which a simple hanging Capacitor Holder was proposed. However since this design had some drawbacks, there was an opening from behind which led to exposure of internal parts to the environment.

3) *Iteration No. 3:*

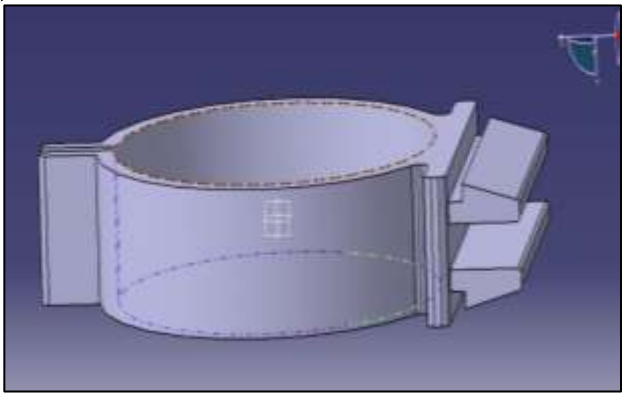
Fig. 3.5: 3rd Iteration of the clamp

Initial design of Snap fit Capacitor holder.

C. Assemblies:

1) Assembly using 1st Iteration:

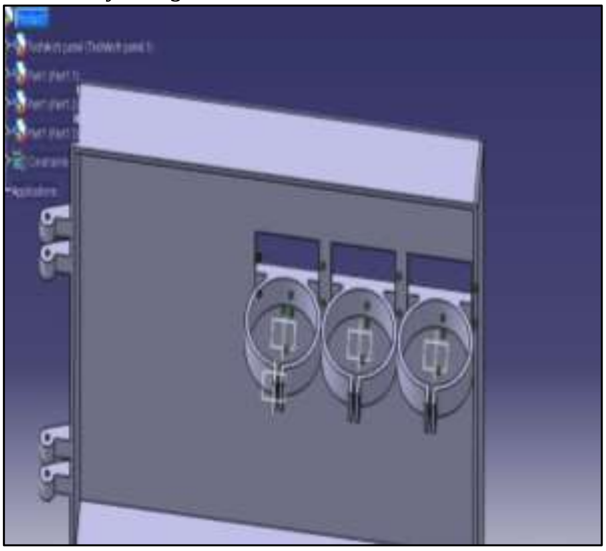

Fig. 3.6: 1st Iteration of part assembly

– Assembly using 2nd Iteration:

In the above Fig. the panel and the capacitor holder are assembled where a simple hanging Capacitor Holder was proposed. But since this design had some drawbacks like, there was an opening from behind which led to exposure of internal parts to environment

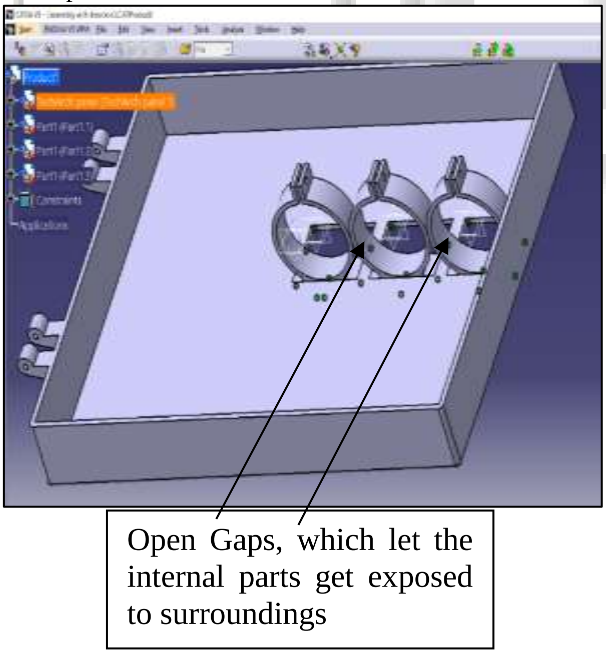

Fig. 3.7: Alternate Design

– Assembly using 3rd Iteration:

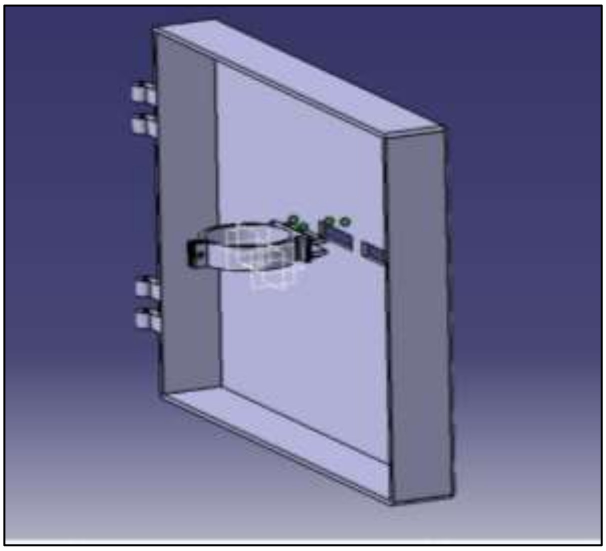

Fig. 3.1: Initial design model.

The initial design contains a snap fit but was designed with a rough idea of Snap fit and minimum research. The holder was initially fit with screws in existing metal panel. The snap replaces the screws reducing the assembly time as well as child part.

(Based on the compliant mechanism the third iteration with holder having snap-fit was proposed. In the above Fig., a panel and Capacitor holder with Snap fit is shown.)

– Assembly of Capacitor:

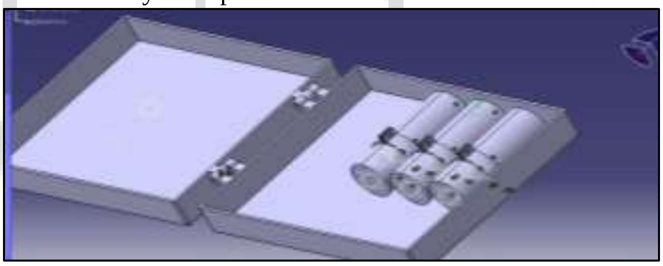

Fig. 3.9: Final Assembly using all parts

In the above Fig. the Assembly of Capacitor, its holder and the Panel is shown. Here a basic snap fit created without considering plastic design consideration.

D. Design Modifications:

Since the snap fit design with further modifying could be designed better post gaining proper knowledge, our team decided to modify the previous design that we modelled in iteration 1. We took help of an industry expert working in the field of plastics and were able to design the snap-fit with proper guidance and knowledge.

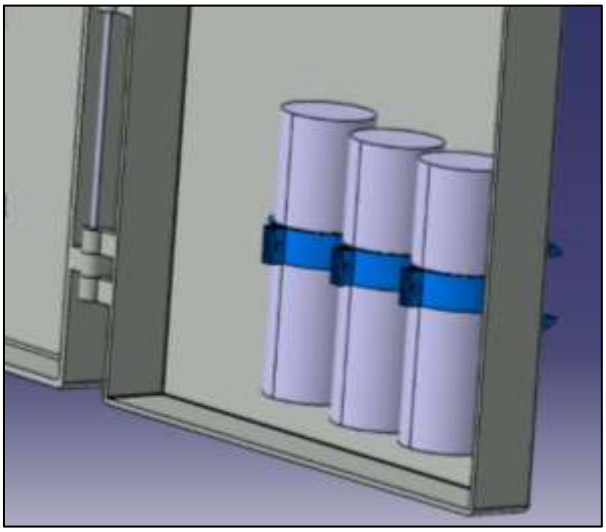

Fig. 3.4: Modified Panel

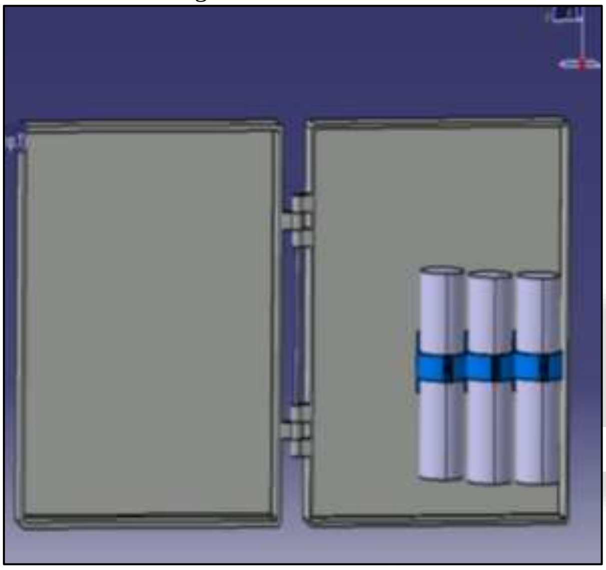

Fig. 3.5: Modified Design

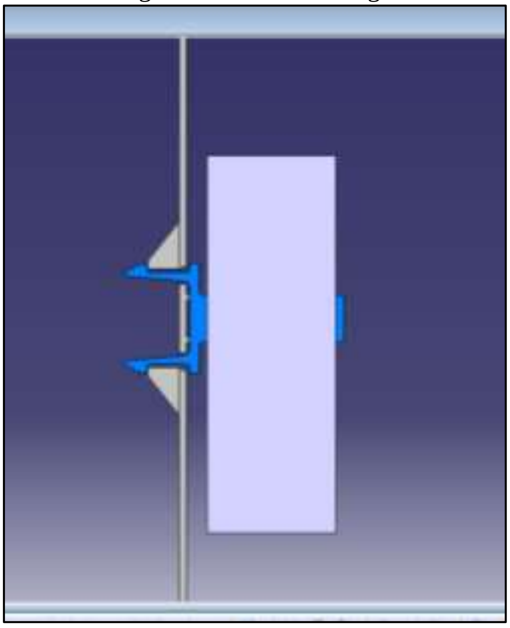

Fig. 3.6: Section View of Modified Snap-fit Design (Without locator)

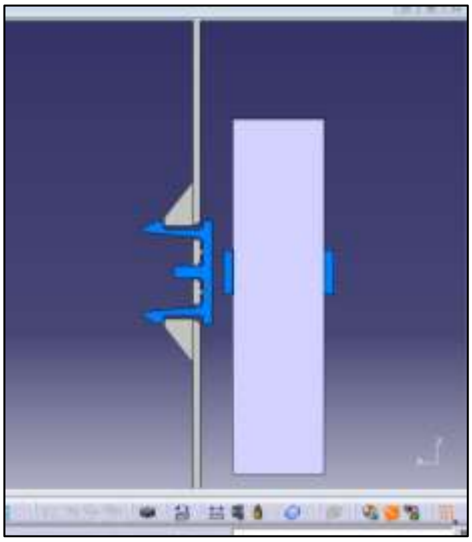

Fig. 3.7: Section View of Modified Snap-fit Design (With locator)

In the Fig 3.6 section view of the final snap fit design is show, where blue colour part is the Holder having snap fit, the Rectangular part (cylindrical in reality) is the Capacitor and the grey colour part is the Panel.

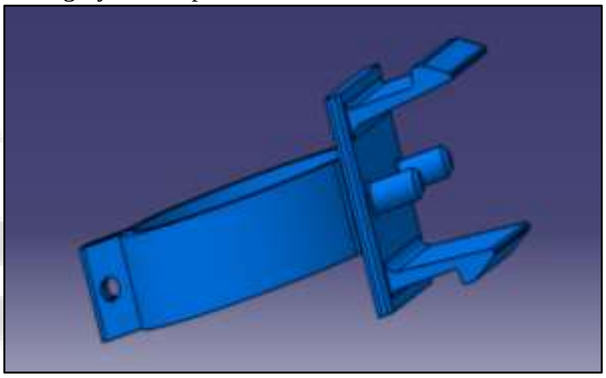

Fig. 3.8: Modified Capacitor Holder

The Holder has a cantilever snap and a locator to restrict its moment up and down.

The Material used for Snap will be PBT, since it has the right amount of flexibility and strength which a snap needs.

E. Design Modifications:

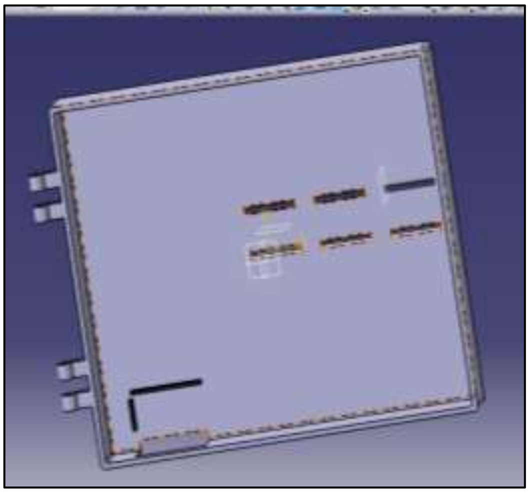

Fig. 3.12: Modified Panel

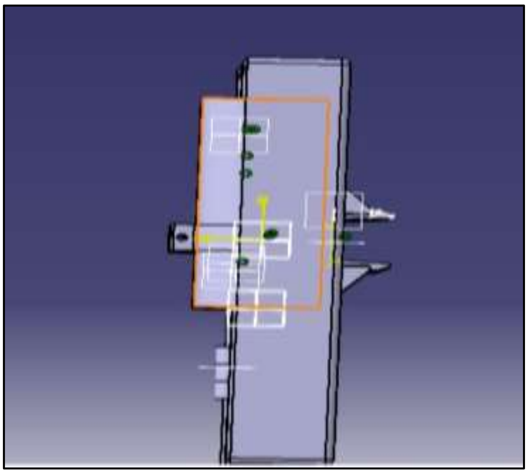

Fig. 3.13: Modified Design

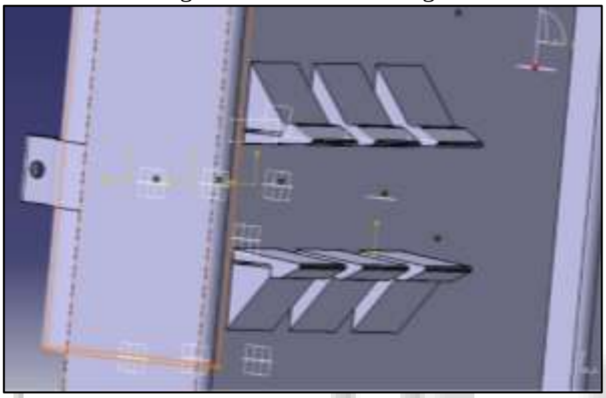

Fig. 3.14: Modified Snap-fit Design

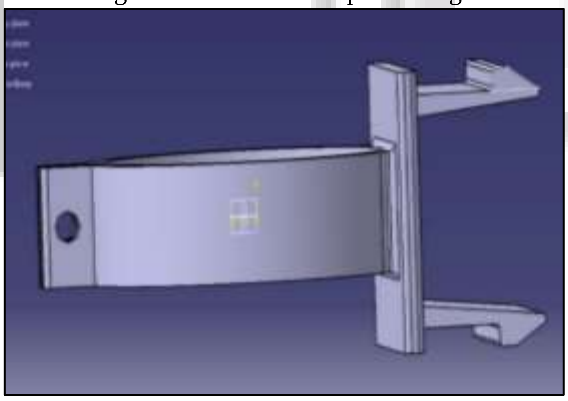

Fig. 3.15: Modified Capacitor Holder

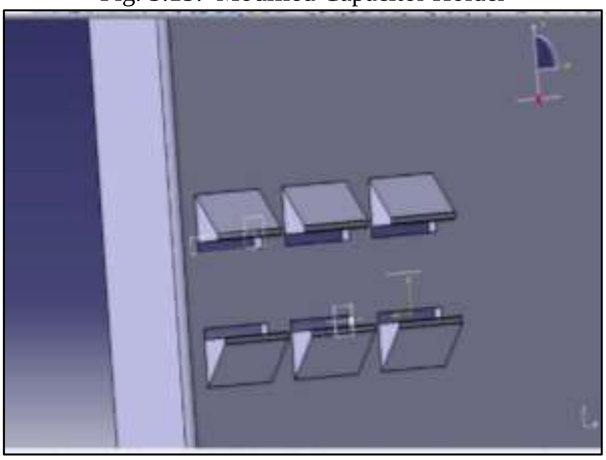

Fig. 3.16: Modified Snap-fit

Final assembly of the Snap-fit design with proper design considerations. Here the snap fit holder, the panel, and the capacitor are shown.

– Advantages of snap fit design
  - Fast assembly.
  - Child parts eliminated.
  - If damaged or in case of maintenance holder can be easily replaced.
– Disadvantages of snap fit design
  - Weaker since plastic is used.
  - It might lose its flexibility over time.

F. Alternate Design for Snap fit design:

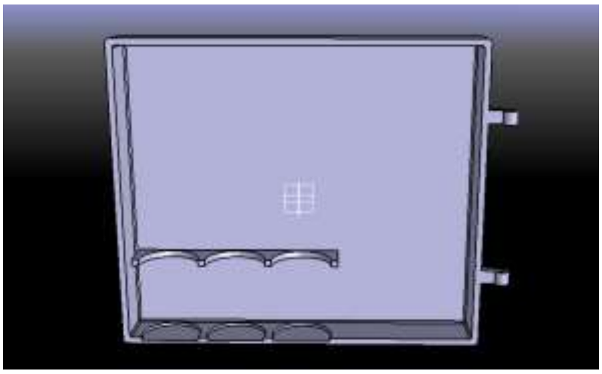

Fig. 3.9: Top Panel with Cylinder

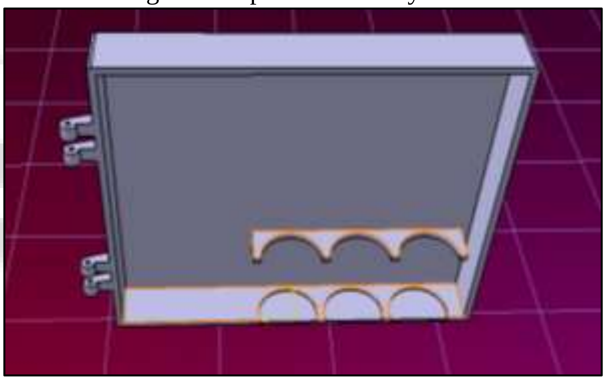

Fig. 3.10: Bottom Panel with Cylinder Holder

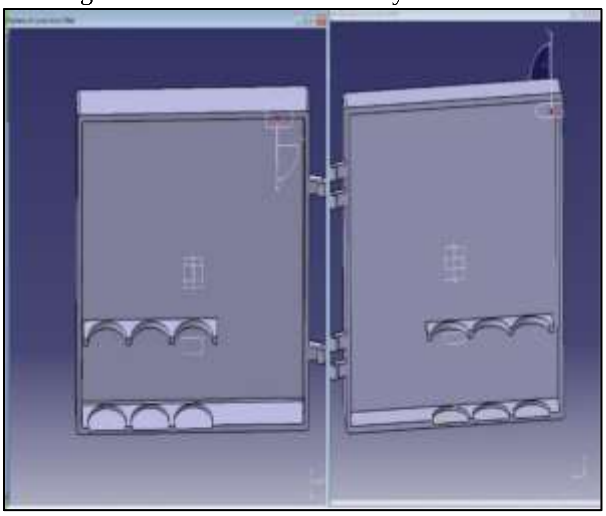

Fig. 3.11: Both panels

– After working on snap fit design, we decided to go for a simpler design in which there will not be any tightening parts.

- We decided to go for a design in which the capacitors will be clamped internally and hence will not require any complicated designs.
- It has a groove on the bottom wall of the panel (which is thickened a little to ensure design safety.
- The groove will hold the capacitor when panel is opened.

To overcome some drawbacks from the snap fit design, such as,

- Snap fit might lose its flexibility over time,
- Eliminating an extra part, i.e., the holder.

The above design was proposed where, the holder was a part of the body itself, this eliminated an extra part and reduced assembly time.

The draw back occurring in the design are,

- If any damage whole part should be replaced.
- Complicated to manufacture

G. Snap-fit Analysis:

1KN force applied

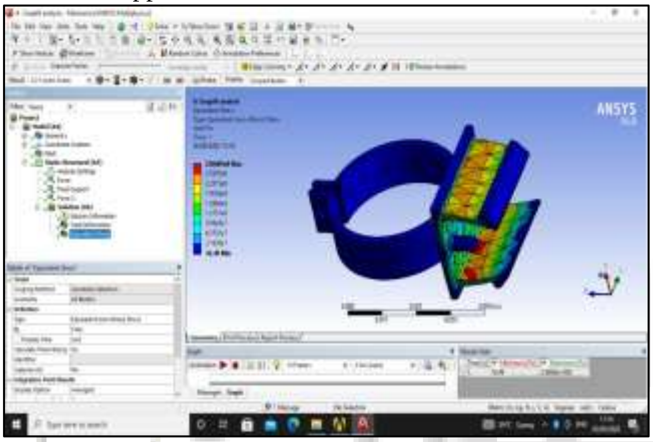

Fig. 3.12: Equivalent Stress for 1KN

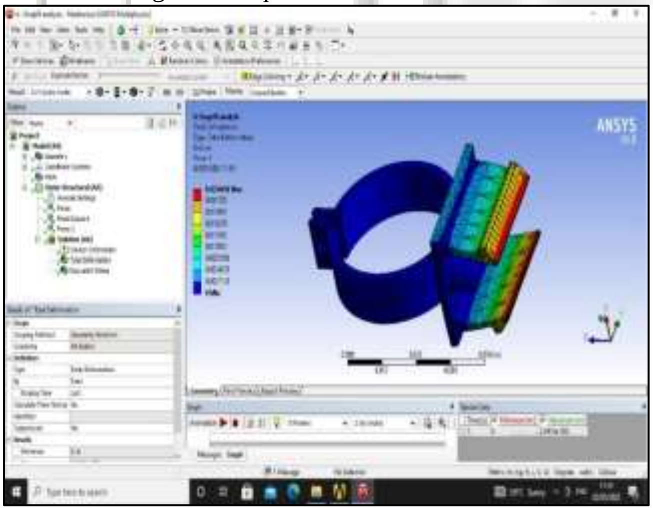

Fig. 3.12: Total Deformation for 1KN

- Material used for this holder is different from the other material for body, which is PBT.
- PBT is used for products which require flexibility and strength together.
- PBT allows the holder to flex while entering the panel and then regain its original shape and hold it.
- Analysis of the snap fit is done for the root and cantilever of the holder.
- The teeth are analysed to find the Force at which it fails.

Force applied is 5KN.

- Stress
  Maximum Stress- 286.9 Mpa
  Minimum Stress- 16.49 Mpa
- Total deformation
  Maximum- 0.0244m
  Minimum- 0m
- 5KN force applied

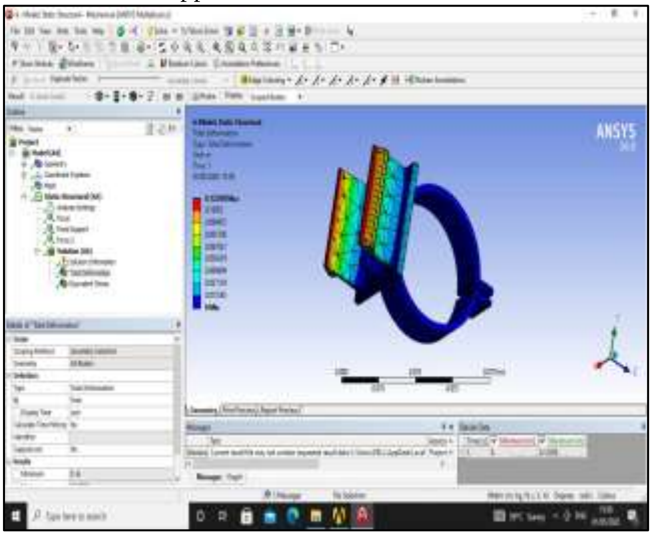

Fig. 3.12: Total Deformation for 5KN

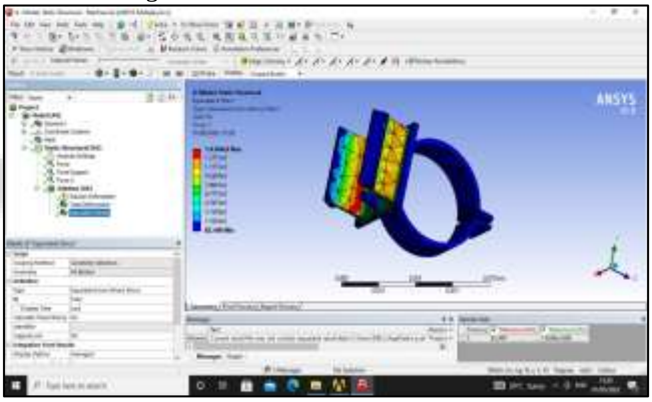

Fig. 3.12: Equivalent Stress for 5KN

- Stress-
  Maximum Stress- 1434.4 Mpa
  Minimum Stress- 82.449 Mpa
- Total deformation
  Maximum- 0.12208m
  Minimum- 0m

H. Design of Mould: CAD model of the Mould:

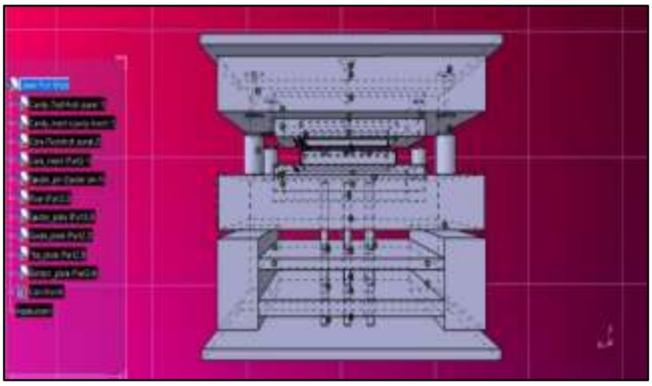

Fig. 4.2: CAD Model of Mould

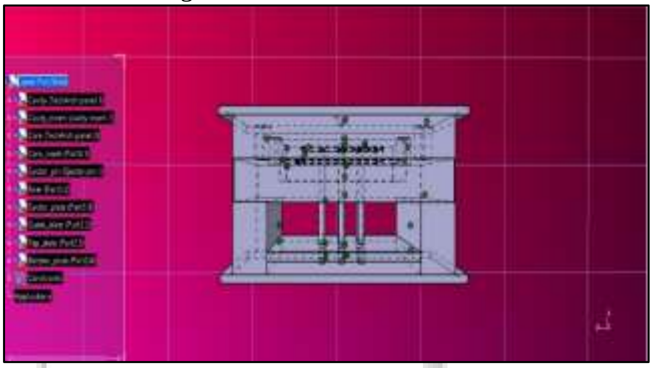

Fig. 4.3: CAD Model of Mould

To understand a basic structure of the mould we designed a mould with the general considerations and keeping into account its general components. This mould was designed keeping into account the simplest design of panel.

IV. COST ESTIMATION:

Parts to be made/month	8000	
Process	Metal Part	Plastic
Powder coating	₹ 56.00	-
Raw Material	₹ 88.00	₹ 93.00
Processing Cost	₹ 26.00	₹ 46.00
	₹ 170.00	₹ 139.00
Total cost	₹ 13,60,000.00	₹ 11,12,000.00

	Per part	For Batch
Cost Savings	₹ 31.00	₹ 2,48,000.00
Savings %	18.24%	18.24%

Table 1: Cost Estimation

V. THERMAL PROPERTIES:

Melting points-
PC-ABS - 130-270 degree Celsius
PBT – 493 degrees Celsius

The panel will be reaching to temperatures of about 70-80 degree Celsius due to internal components like capacitors motor control unit.

Therefore the material used is safe in terms of thermal properties since it can handle the temperature point that the internal components reach to.

VI. CONCLUSION:

The Metal panel redesigned with Plastic (PC ABS and PBT) was proposed.The following objectives were achieved.

– Cost reduction up to 20%
– Reduced child parts such as nuts and bolts and consequently resulting to Assembly time reduction
– A motor panel designed to survive in environmental conditions.
– Using plastic the panel was made electrically non-conductive, thus was made shock resistant and safe to use near water.

REFERENCE:

[1] Engineering Plastics- Part and Mould Design- A Design Guide.
[2] Problem occur During Injection Moulding Parts by- Rushikesh D. Dhomne, Abhijeet A. Raut, Kalyan labade Student Assistan Professor Industry Supervisor Department of Mechanical Engineering G. H. Raisoni College of Engineering, Nagpur, India S. P. Auto Engineering Pune, India.
[3] https://www.stratasys.com
[4] https://www.researchgate.net/publication/265224787.
[5] https://www.researchgate.net/publication/343788280.
[6] Research paper by- Plastics engineering company sheboygan , wisconsin 53082 – 0758 U.S.A
[7] A review of Injection moulding process on the basis of runner system and process variables by- Dr. Prachi Kale Ph. D Research scholar, at G H Raisoni College of Engineering and Management, Pune.