Available online www.ejaet.com

European Journal of Advances in Engineering and Technology, 2015, 2(3): 90-95

Research Article **ISSN: 2394 - 658X**

A Design and Moulding Analysis of Two Plate Mould Tool for Motor Rare Housing Thermoplastic Product

Raghavendra N[1], Pramod R Sharma[2] and M C Math[3]

[1,2] *MVJ College of Engineering, Bangalore, India*
[4]*Visvesvaraya Technological University (VTU) PG Centre, Mysore, India*
nraghavendra85@gmail.com

ABSTRACT

This deals with design and analysis of two plate mould injection mould tool. The objective of this is to design two plate mould tool and analyze the material flow of the selected product. This describes the material flow analysis to analyzed the material flow in the part and identify the best gate locations of the components. The structural three-dimensional solid modeling of two plate mould was developed using the computer-aided drawing software. The strategy of analysis model was developed. The material flow and gate location analysis was then performed using Mould Flow low Plastic Insight software. The analysis model of the component was analyzed using the single cavities with cold runner design. It is observed that the analysis using Mould flow material flow analysis show the different orientation skin for different gate locations. Based on the analysis, improvement has been made to the part design and feed systems in the mould. This includes the location of the gate at the part design, size of sprue, runners and gates. From the analysis also, Plastic insight5 software helps to determine the part defects that might occurred during plastic injection moulding process such as Short shot, unequal filling, over filling, welding lines and others. From that, the optimum parameters setting are selected in order to get a quality plastic. So this will explain the plastic flow analysis clearly. The final design was done in solid modeling and details drawing using Solid Works software. The results can also significantly reduce the cost and time to fabricate the Tool. That is also can be a reference for the future works of fabrications.

Key words: Injection mould design, weld lines, fill time, air traps, and tool design

---

INTRODUCTION

Plastic is one of the most versatile materials in the modern age which is widely used in many products in different shapes which are moulded through the application of heat and pressure .Injection moulding has become the most important process for manufacturing plastic parts due to its ability to produce complex shapes with good dimensional accuracy. However, the current plastics industry is under great pressure, due to the globalization of the market, and high demand of product. In injection moulding, the design of mould is of critical importance for the product quality and efficient processing, which is also responsible for the economics of the entire process. Mould designers are required to possess thorough and broad experience, because the detail decisions require the knowledge of the interaction of various parameters [1]. Due to the lack of experienced designers, intelligent CAD tools that can assist in the various tasks of the mould design process can be used for the good productivity of mould making industry. The injection moulding process involves feeding raw material, plasticize the raw material, fill the mould, pack the mould, hold pressure, cooling of mould and lastly opening of mould and part ejection.

The main factors in the injection moulding are the temperature and pressure history during the process, the orientation of flowing material and the shrinkage of the material. The raw material is generally fed through an augur type sprue channel which feeds the resin pellets forward inside the heated barrel. The resin fed in the barrel through the feed throat is fixed by screw extruder. The heater bands surrounded on barrel maintains the melt temperature of the resin. This plasticized resin is then injected into the mould after mould closing. Injection of melt is generally done into the cooled mould by hydraulic mechanism [2]. This filling phase generally takes few seconds depending on plastic grades, wall thickness, and the shape of the part. After filling the mould cavity with resin the pressure is reduced to the pack value and maintain for a specified time to assure the mould is full. Thereafter a hold pressure is maintained for a set period of time on the solidifying material within the mould. This holding pressure is only effective as long as gate remains open.

DESIGN METHODOLOGY

This paper highlights a practical design procedure/ methodology of an injection moulding die, adopted by analyzing the various parameters to produce a precession industrial component namely RAM component for an electrical transformer has been choked out in detail aspect which is as follows. To start up a new mould design, the designer should know some important points to avoid some mistakes before going further. i.e., Product outlook design, material usage, correction shrinkage of the material, number of cavities and selection of mould base [3]. In injection moulding, there is an optimum gate size and it should large enough for suitable fill rate and small enough seal off and prevent back flow or over packing. Fig.1 Shows the Standard Two Plate Mould. The mould which contains basic parts

a) A stationary mould half on this side where plastic is injected
b) A moving half on the closing or ejecting side.

The separation of the two moulds is called the parting line. Usually the melted plastic material is fed through the central feed channel called sprue of the cavity. The mould will coincide with machine cylinder nozzle. Up design for the plastic part and mould design process and reduce the long lead time.

CAD/CAM FOR MOULD DESIGN

The architecture of injection mould design system is proposed based on practical design parameters and conceptual design stage mainly consists of concept generation and concept evaluation. CAD/ CAM can help designers to speed up design for the plastic part and mould design process and reduce the long lead time

1. The mould has to be designed to produce good quality FRP component considering the ease of manufacturability, assembly and positive ejection of the component within the minimal time and cost [4]. The design of single cavity moulding tool is carried out using software PRO-E.
2. The mould flow analysis using AUTODESK MOULD FLOW software is carried out for the component in order to achieve a good quality mould before moulding and to check the manufacturability of plastic part.
3. The detailed drawing of mould is prepared by AUTO CAD and it is used for manufacturing the tool.
4. The mould is manufactured as per drawing specification.
5. The performance of tool is tried out and subsequently the defects are troubles hooted.

Fig.2 shows the Flowchart for practical tool design and Manufacture. The flow chart of the methodology followed for the manufacturing of mould design is as follows.

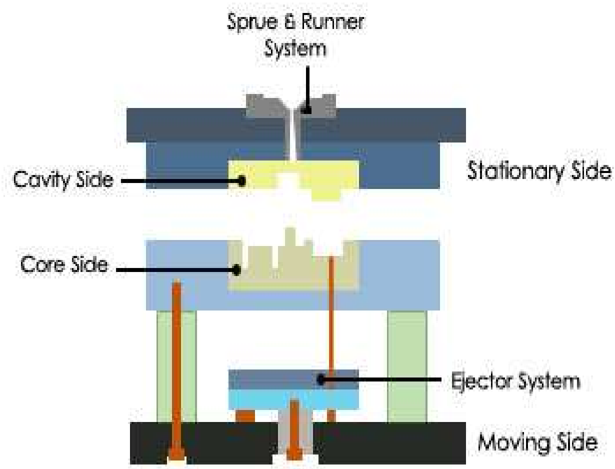

Fig.1 Standard Two Plate Mould [2]

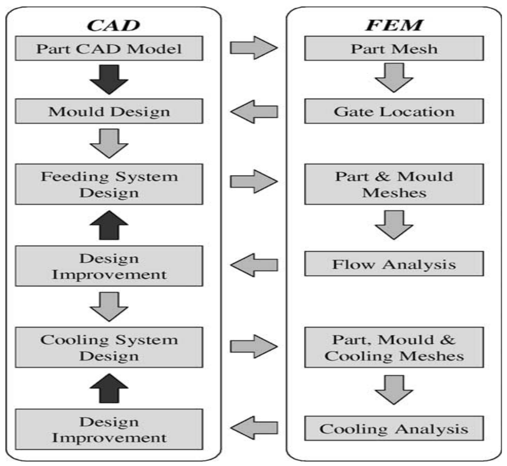

Fig.2 Flowchart for practical tool design and Manufacture [3]

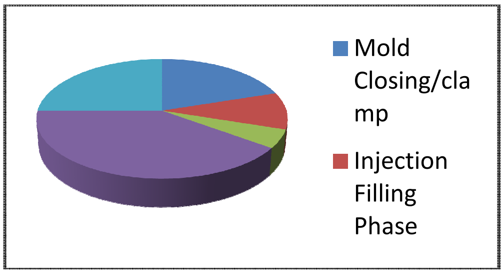

Fig. 3 Injection mould cycle

INJECTION MOULD CYCLE

The time from the moment the mould is closed for one injection, until it is closed for the following injection is termed as cycle time. Typical cycle time ranges from 20 sec to 60 sec which leads to a high production rate with good dimensional tolerance control. The injection moulding process basically involves three phases are Filling phase, Pressure phase, Cooling phase. Fig. 3 Shows the Injection mould cycle.

Injection moulding is the most commonly used process to realize plastic parts with high production rates and good control on the product dimensions. This cyclical process is carried out in three phases: (i) filling, (ii) packing and (iii) cooling. During the first phase, a melted polymer fills the part cavity, moving through the sprue, runners and gates. In the packing phase, additional melted polymer enters the cavity to balance the part shrinkage caused by cooling. The cooling phase takes place concurrently with the filling and packing phases and considers polymer plasticization and the additional time required to obtain a more than 80% solidified product. This is necessary for the ejection of the part which together with mould opening completes the process. During these phases, interactions between material properties, machine parameters and process variables make fabrication complex.

Component Details

Component name: Rare Housing
Component material: Polyamide (Nylon 66) PA66
Shrinkage: 1.5 %
Component weight: 10.5 grams
Moulding type: Single Cavity injection mould tool

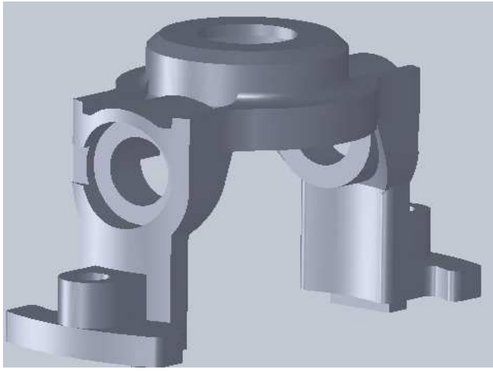

Fig. 4 3D Component Details [2]

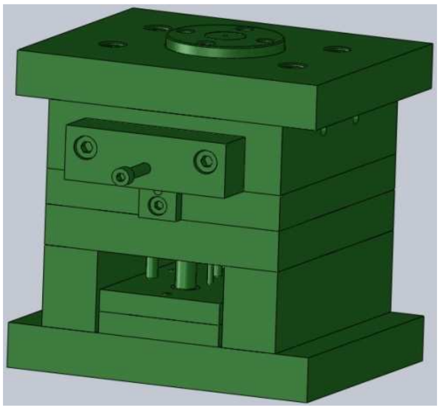

Fig. 5 Full Tool assembly [2]

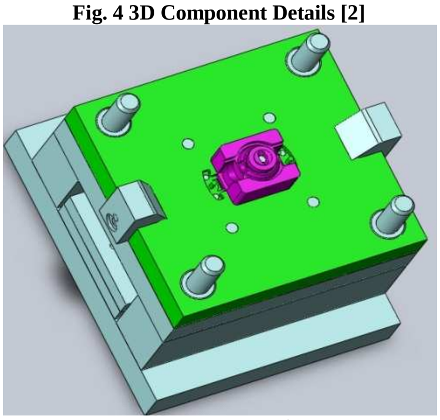

Fig. 6 Moving Half [2]

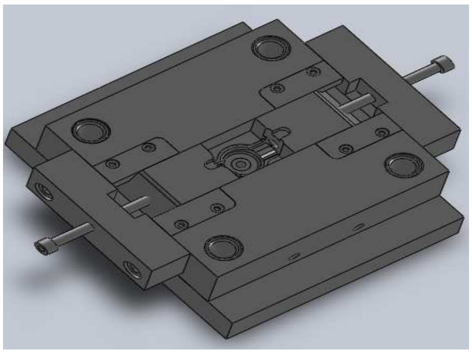

Fig. 7 Fixed Half [2]

NEED FOR SIMULATION

The production of injection moulded parts is a complex process where, without the right combination of material, part and mould design and processing parameters, a multitude of manufacturing defects can occur, thus incurring in high costs. Optimal setting up of injection moulding process variables plays a very important role in controlling the quality of the injection moulded products [4]. Computer aided engineering (CAE) tools can be used to simulate a wide variety of phenomena that occur during the manufacture of plastic parts. The simulation of mould filling in injection process has been presented [5].

The simulation results can be used to correct the defects on the final part, for example by adjusting process settings or modifying the mould design. Injection moulding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost. The cost of tooling for injection moulds can be very high and subsequent rework increases these already high costs. All these factors combine to make injection moulding an ideal application for CAE simulation, the benefits of which provide users with a high return on investment. Computer-aided engineering (CAE) is the assistant to process and calculate the plastic material flow inside the injection moulding.

MOULD FLOW ANALYSIS

The mould flow analysis was performed using Plastic Insight Mould Flow analysis software. The sequence of work involved in mould flow analysis is given below.

- 3D model is converted into STEP format.
- Model meshing is done using dual domain type of mesh.
- Importing the meshed file to the solver package specifying the boundary condition, loads such as injection pressure, injection time, mould temperature, melt temperature, material properties etc. Building feed system such as sprue, runner and gate, cooling lines. Mesh the feed system and cooling lines. Run the analysis for different analysis types like fill, flow, cooling, war page etc. Study the result, interpret them.
- Establish the optimized data for runner, gate, sprue dimensions, coolant temperature etc.

Based on the analysis the optimal combination of part geometry, material choice, best gate location and process parameter to produce quality finish part are determined. This analysis also gives the result of fill analysis, pack analysis, warp analysis and cooling analysis.

Fill Analysis

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each colour contour represents the parts of the mould which were being filled at the same time. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no colour. Fill time is the time taken to fill up the part inside the cavity; it is also to show how the plastic material flows to fill the mould. From that we know that the short shot (part of the model which did not fill) part will be displayed. From that result one can also understand how the weld line and air trap will form. Fig. 8 shows the material filling into the mould.

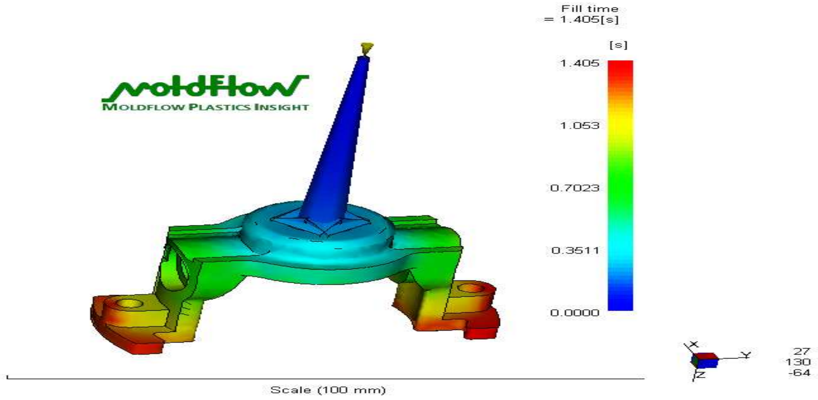

Fig. 8 Fill time plot

Injection Pressure

The Pressure result is generated from a Fill analysis, and the Fig. 9 shows the pressure distribution through the flow path inside the mould at the time the result was written. The pressure difference from one location to another is the force that pushes the polymer melt to flow during filling. The pressure gradient is the pressure difference divided by the distance between two locations. Polymer always moves in the direction of the negative pressure gradient, from higher pressure to lower pressure [6]. (This is analogous to water flowing from higher elevations to lower elevations). Thus, the maximum pressure always occurs at the polymer injection locations and the minimum pressure occurs at the melt front during the filling stage. The colour at each place on the model represents the pressure at the place on the model. Two colours show the highest pressure (red) and lowest pressure (blue).

The injection pressure can be used in conjunction with pressure drop result. For example, even if a section of a part has an acceptable pressure drop, the actual injection pressure in the same area may be too high. High injection can cause over packing.

Temperature at Flow Front

Fig. 10 shows the Temperature at flow front. The flow front temperature result uses a range of colours to indicate the region of lowest temperature (colour blue) through to the region of highest temperature (colour red). The colors represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.

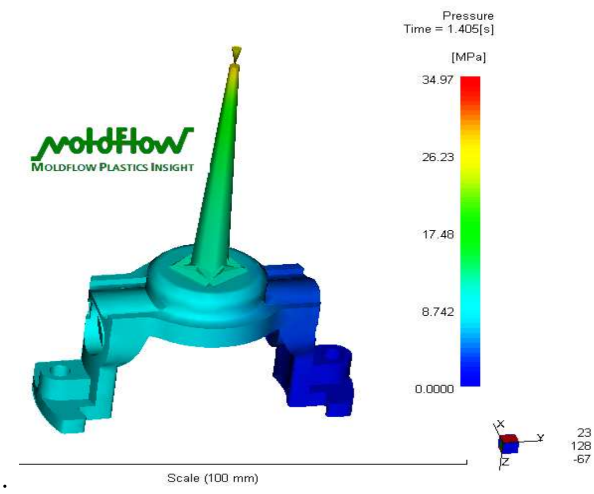

Fig. 9 Injection Pressure distribution

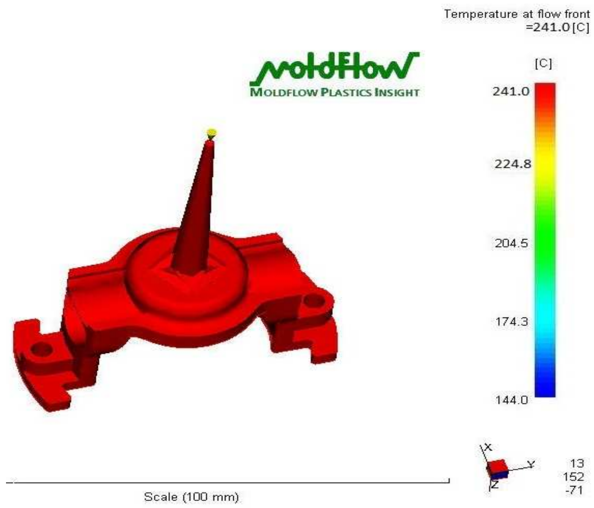

Fig. 10 Temperature at flow front

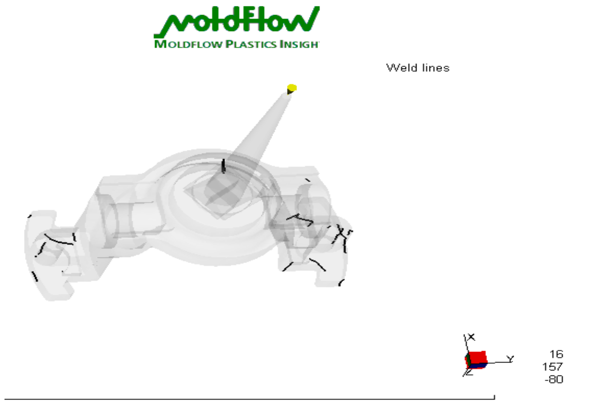

Fig. 11 Weld Lines result

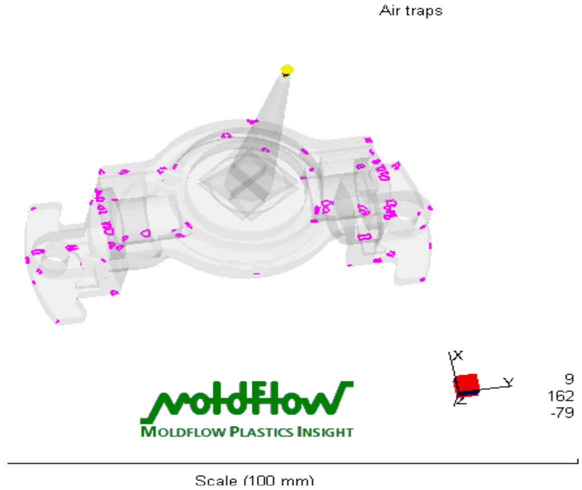

Fig. 12 Air Traps result

Weld Line Result

Fig. 11 shows the weld lines result shows the angle of convergence as two flow fronts meet. The presence of weld lines may indicate a structural weakness and/or a surface imperfection. The term weld line is often used to mean both weld and meld lines. The only difference between a meld line and weld line definition is the angle at which they are formed. Weld lines are formed at lower angles. Weld lines can cause structural problems, and they can also make the part visually unacceptable.

Therefore weld and meld lines should be avoided if possible. However, weld lines are unavoidable when the flow front splits and comes together, around a hole, or has multiple gates. Look at the processing conditions and the weld line position to decide if the weld lines will be of a high quality. Avoid weld lines in areas which need strength, or which need to appear smooth.

Fig. 12 shows the Air traps result a thin, continuous line wherever an air trap is likely to occur. An air trap is where melt traps and compresses a bubble of air or gas between two or more converging flow fronts, or between the flow front and the cavity wall. Typically, the result is a small hole or a blemish on the surface of the part. In extreme cases, the compression increases the temperature to a level that causes the plastic to degrade or burn. Move the Injection location so that the air traps form in easy-to-vent areas, such as at the parting plane.

WARP Analysis

This warp analysis results show the deflection at each node of the component along x, y and z axis. Fig. 13 shows the Warp Analysis. This highlights the effects of differential shrinkage, deflection at each node, effect of differential cooling and corner effects on the deflection.

Best Gate Location

The analysis is for predicting the Best Gate Location, shown in dark blue. Fig. 14 shows the Best gate location. Red colour indicates the worst location.

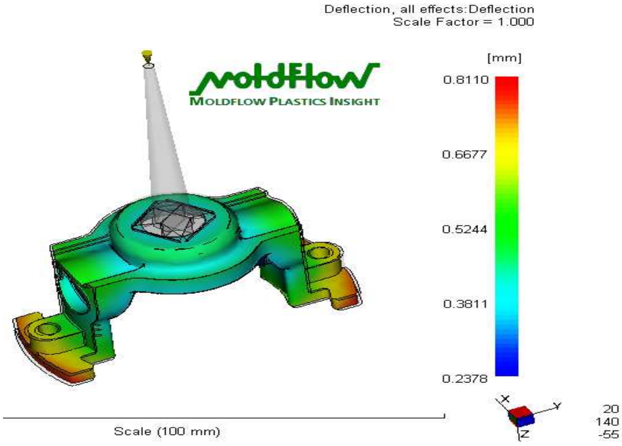

Fig. 13 Warp analysis

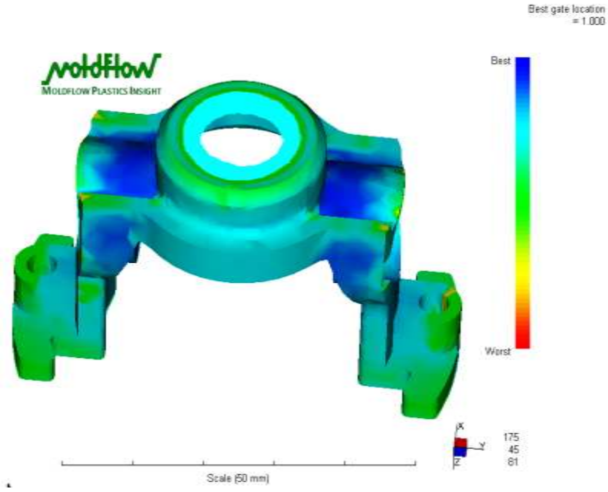

Fig. 14 Best gate location

CONCLUSION

- Best location of the gate obtained from the analysis has been selected for manufacturing.
- The present gate size and location is finalized after balanced filling and other factors.
- The temperature at flow front in the most region of the component varies from 210^{o}C to 2410^{o}C which is close to the melt temperature of the polymer binder (2300^{o}C), so formed weld line has little effect on the strength of the product.
- Most of the air traps are located on the surface, which can be removed by providing suitable air vents in the mould.

REFERENCES

[1] PK Bharti and MI Khan, Recent Methods for Optimization of Plastic Injection Moulding, *International Journal of Engineering Science and Technology*, **2010**, 2(9), 4540-4554.

[2] Technical Directory on Design and Tooling for Plastics, *CIPET, Mysore, Karnataka, India* (2012).

[3] RGW Pai, *Injection Mould Design*, 4th Edition, Affiliated East-West Press Pvt, **2014**.

[4] D Mathivanan, M Nouby and R Vidhya, Minimization of Sink Mark Defects in Injection Moulding Process - *Taguchi Approach*, *International Journal of Engineering, Science and Technology*, **2010**, 2 (2), 13-22.

[5] J Nabialek and J Koszkul, Model Optimization for Mould Filling Analysis with Application CAE Package C-Mould, *Journal of Achievements in Materials and Manufacturing Engineering*, **2006**, 19 (1), 75-82.

[6] Wong CT, Shamsuddin Sulaiman, Napsiah Ismail & AMS Hamouda, Design and Simulation of Plastic Injection Moulding Process, *Pertanika J, Sci &Techno Supplement*, **2004**,12 (2), 85-99.