Optimizing the Die Design Parameters for FRP Components Produced in Injection Molding using Mold Flow Analysis

Dr.J. Fazlur Rahman*, Mohammed Yunus, Mohammed Irfan***, T.M. Tajuddin Yezdani***

*(Professor Emeritus, Department of Mechanical Engineering, H.K.B.K. C.E., Bangalore, Karnataka State, India-45

** (Professor, Department of Mechanical Engineering, H.K.B.K. C.E., Bangalore, Karnataka State, India-45

*** (Lecturer, Department of Mechanical Engineering, H.K.B.K. C.E., Bangalore, Karnataka State, India-45

*** (Professor, Department of Mechanical Engineering, H.K.B.K. C.E., Bangalore, Karnataka State, India-45

ABSTRACT

Injection molding is the most widely used method for the production of intricate shape plastic parts with good dimensional accuracy. The current plastic industry is under great pressure due to the globalization of the market, the short life cycle of the product, increasing diversity and also the high demand of product quality. To meet such requirements it is very important to adopt various advanced technologies like CAD/CAE/CAM, concurrent engineering and so on for the development of the injection molded part. The design of a mold is of critical importance for product quality and for efficient processing, it is also an integral part of plastic injection molding as the quality of final plastic part is greatly reliant on injection mold. A two plate injection mold design and mold flow analysis of single cavity injection mold for a given component was taken up according to the customer's specification. The material selected for the RAM component was Polypropylene with 20% Glass fiber. The design of the tool includes the design of core insert, cavity insert, cavity replicable-inserts & side-core assembly. After thorough study of the component for shape, size, critical parameters, to help in proper selection of parting surface, ejection system, feeding system, selection of suitable injection molding machine based on clamping force calculation and the manufacture of mold to produce better-quality components was done. The 3D Virtual Model of the tool and the core-cavity extraction was performed using the Pro-Engineer Wildfire-4. The 2D manufacturing drawings were drafted in *Autocad-2007*. Mold flow analysis was performed using Auto desk mold flow analysis software to determine the optimal combination of part geometry, material choice, the best gate location and process parameters to produce quality finished parts. This analysis also gives the picture of fill analysis, pack analysis, warp analysis and cooling analysis. With the help of the manufactured mold produced based on the analysis, was tested for the production of final product, with precise specification of the customer's requirement. This work can also be further extended for a detailed stress and fatigue analysis for the core and cavity inserts to obtain better tool life.

Keywords - Injection molding, Die Designing Methodology, Mold Flow Analysis, Problems and Causes, solutions and Troubleshooting.

I. 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 molded through the application of heat and pressure [2]. Injection molding has become the most important process for manufacturing plastic parts due to its ability to produce complex shapes with good dimensional accuracy [3]. However, the current plastics industry is under great pressure, due to the globalization of the market, and high demand of product. In injection molding, 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.

Mold designers are required to possess thorough and broad experience, because the detail decisions require the knowledge of the interaction of various parameters. Due to the lack of experienced designers, intelligent CAD tools that can assist in the various tasks of the mold design process can be used for the good productivity of mold making industry [4]. The injection molding 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 [14].

The main factors in the injection molding 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 [6]. The resin fed in the barrel through the feed throat is plasticized 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 mold by hydraulic mechanism. 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 mold is full. Thereafter a hold pressure is maintained for a set period of time on the solidifying material within the mold. This holding pressure is only effective as long as gate remains open. Critical dimension, surface finish, cycle time etc are all affected by mold cooling [16]. Hence mold cooling is a decisive factor for producing components with good quality and at competitive cost. And as the cooling phase is completed the mold is opened and the component is ejected with ejector pins of the tool. Once the molding is clear from the molding tool, the complete molding cycle is repeated [17].

II. DESIGN METHODOLOGY

This paper highlights a practical design procedure/ methodology of an injection molding die, adopted by analyzing the various parameters to produce a precious industrial component namely RAM component for an electrical transformer has been choked out in detail aspect which is as follows.

2.1. Design Methodology

To startup a new mold 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 mold base. In injection molding, 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 [15]. The mould which contains one or two basic parts

1. A stationary mould half on this side where plastic is injected
2. 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.

2.2. 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 [20].

1. The mold 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 [22]. The design of single cavity molding tool is carried out using software PRO-E.
2. The mold flow analysis using AUTODESK MOLD FLOW software is carried out for the component in order to achieve a good quality mold before molding and to check the manufacturability of plastic part.
3. The detailed drawing of mold is prepared by AUTO CAD and it is used for manufacturing the tool.
4. The mold is manufactured as per drawing specification.
5. The performance of tool is tried out and subsequently the defects are troubleshooted.

The flow chart of the methodology followed for the manufacturing of mold design is as follows

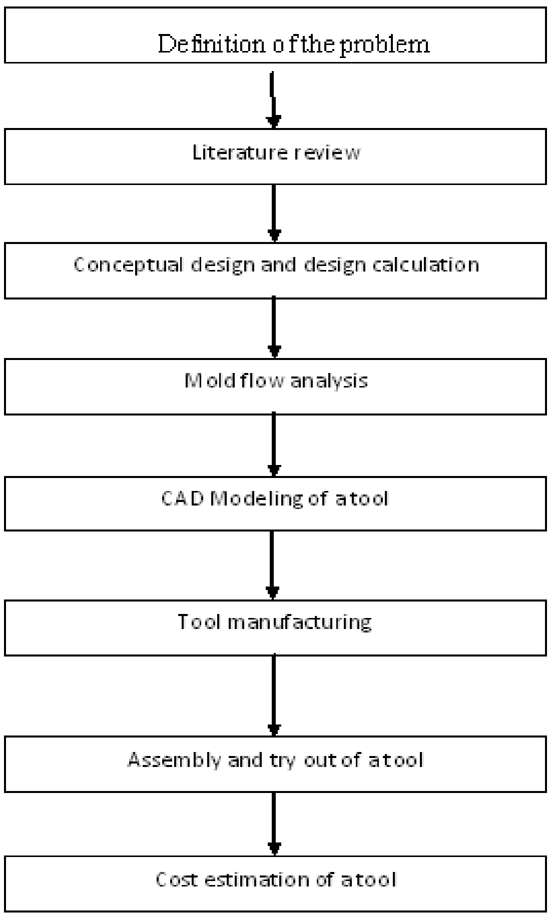

Fig.1. Flowchart for practical tool design and Manufacture

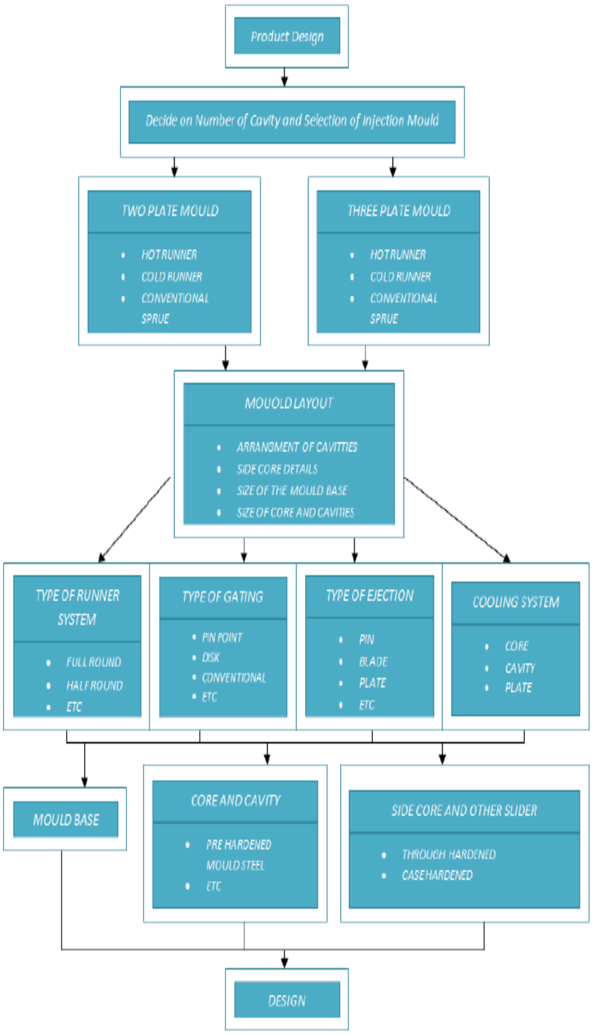

Fig.2. Flow Chart of Mold Design Procedure

For the complete design of mold the following step by step procedure is adopted [15]

1. Study of component and its specification: Material. Density, shrinkage, volume, thickness etc., to be known.
2. Weight of component: The Weight of the component including losses due to shrinkage of plastic material, weight of sprue and runner. The actual weight of component with some losses was calculated by multiplication factor based on the actual weight.
3. The minimum wall thickness of core and cavity insert is also calculated using the standard formulae.
4. The standard mold base dimensions are selected according to DME.
5. The clamping force required is calculated based on the values of injection pressure, total projected area of the component and the molding.
6. The shot capacity of machine is calculated based on plasticizing rate, heat content and total heat of the plastic material.
7. The number of cavity based on either shot capacity or plasticizing capacity, is computed and selected whichever is higher.
8. The design of feeding system which involves runner design and gate design are computed with the knowledge of the weight of the component and the length of the runner.
9. The cooling system (weight of water to be circulated per hour and to dissipate the heat) is calculated after finding out the heat transfer from mold per hour.
10. The design of ejector system which involves ejector pin size and number of ejector is also decided based on bearing surface area of the component.
11. Lastly, the selection of guide pillar and guide bush with their dimensions are chosen according to DME standard.

The critical items to be considered while designing an injection mould tool are shrinkage, draft angle, selection of parting surface, number of cavities, feed system, cooling system, ejector system and venting.

The 3D virtual model of the tool (with the design dimensions) and the cavity extraction is performed using PRO-E and 2D detailed drawing are drafted in AutoCAD for the manufacturing the tool.

2.3. Mould Construction

In the design of injection mould, the following machine specifications are also to be considered [20], [21].

1. Injection pressure
2. Shot weight
3. Clamping force
4. Distance between tie bars
5. Min and max Mould height
6. Mould opening stroke
7. Ejection force
8. Maximum ejection stroke.

III. Mould Flow Analysis

The mold flow analysis was performed using Autodesk Mold Flow analysis software.
The sequence of work involved in mold flow analysis is given under.
1. Converting the 3D model in STEP format.
2. Meshing the model by using dual domain type of mesh.
3. Importing the meshed file to the solver package specifying the boundary condition, loads such as injection pressure, injection time, mold temperature, melt temperature, material properties etc.
4. Building the feed system such as sprue, runner and gate, cooling lines.
6. Mesh the feed system and cooling lines.
7. Run the analysis for different analysis types like fill, flow, cooling, warpage etc.
8. Study the result, interpret them.

9. 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.

3.1. Fill Analysis
3.1.1. Fill Time

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each color contour represents the parts of the mold 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 color. 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. Figure 3 shows the material filling into the mould.

Processing parameters

Injection pressure	-100 MPa	
Recommended Melt Temperature	-	200°C
Recommended Minimum Mold Temperature	-	65°C

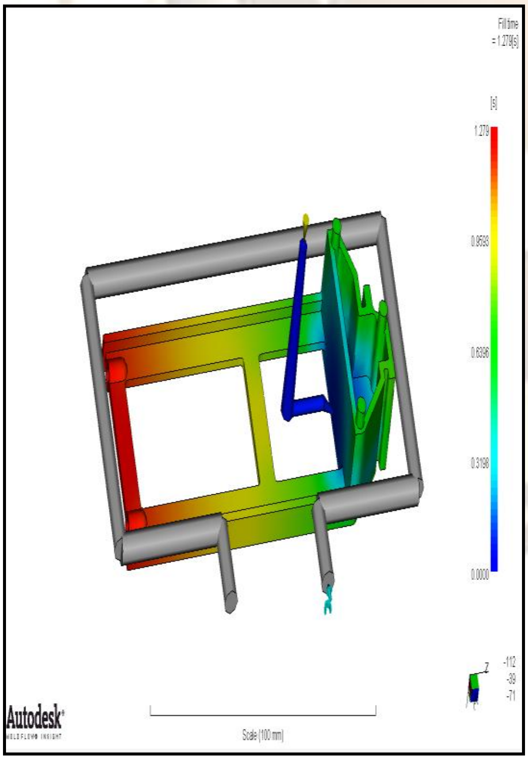

Fig.3. Fill time plot

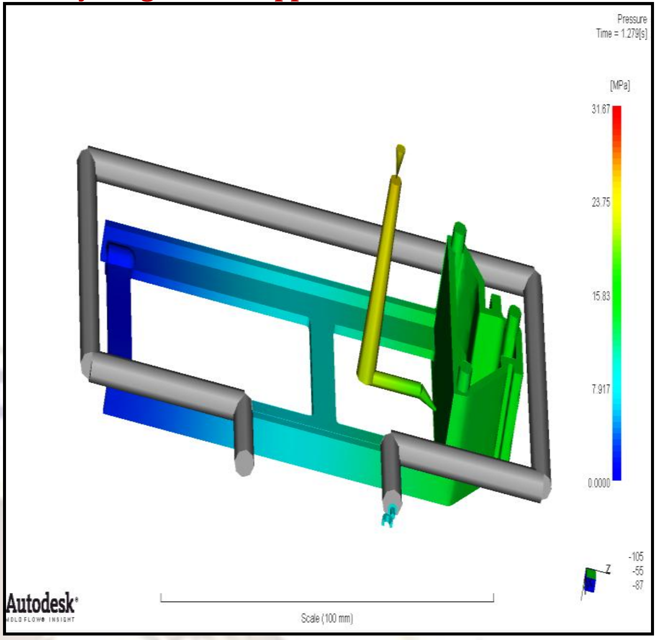

Fig.4. Injection Pressure Distribution

3.1.2. Injection pressure

The Pressure result is generated from a Fill analysis, and the Figure 4 shows the pressure distribution through the flow path inside the mold 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. (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 color at each place on the model represents the pressure at the place on the model. Two colors 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. To reduce the chance of this happening, follow procedure is followed:

1. Increase the maximum injection pressure
2. Alter the polymer injection location
3. Alter part geometry
4. Select a different material

3.1.3. Weld lines result

The Weld Lines result in Figure 5 displays the angle of convergence as two flow fronts meet. The presence of weld lines may indicate a structural weakness and/or a surface blemish. 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.

3.1.4. Poisson's Ratio (fiber) Result

The Poisson's ratio (fiber) result is a fiber-over-thickness (FOT) averaged value, therefore, it is an element-based value averaged over all the laminates of each element. This result is generated from a Fiber orientation Pack analysis and it is shown in Figure 6 The Poisson's Ratio (fiber) result indicates the average strain in the second principal direction caused by the stress in the first principal direction. The Poisson's ratio, (v12) result is a laminate-based value; therefore, you can view the Poisson's ratio distribution for each laminate in the model. We can check this result for more detailed information on the Poisson's ratio distribution. Poisson's ratios are mechanical property values. The distribution of this mechanical property is used by the structural analysis for its performance evaluation in a Stress analysis.

3.1.5. Sink Marks and Index Result

The Sink marks, index is an indication of the potential shrinkage due to a hot core. It is calculated for each element at the instant when local pressure has decayed to zero during the packing stage, and reflects how much material is still melt and left unpacked. Higher Sink marks, index value shows higher potential shrinkage. However, whether or not the shrinkage would result in sink mark depends on geometry characteristics.

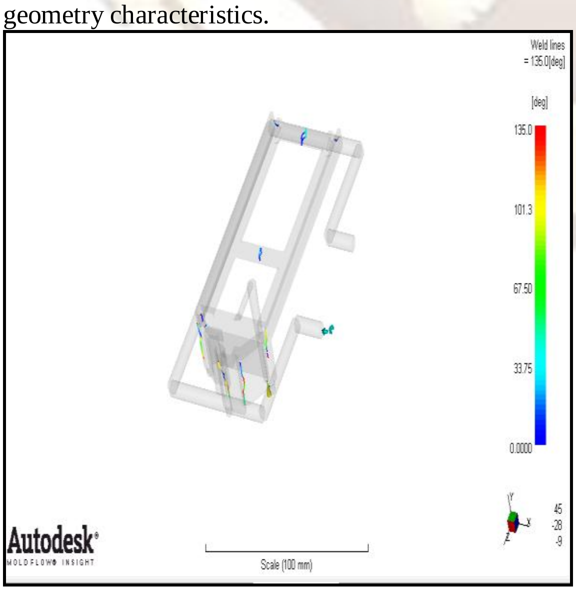

Fig.5. Weld line plot

The Sink marks, index generated indicates the likely presence and location of sink marks (and voids) in the part. This is shown in Figure 7.

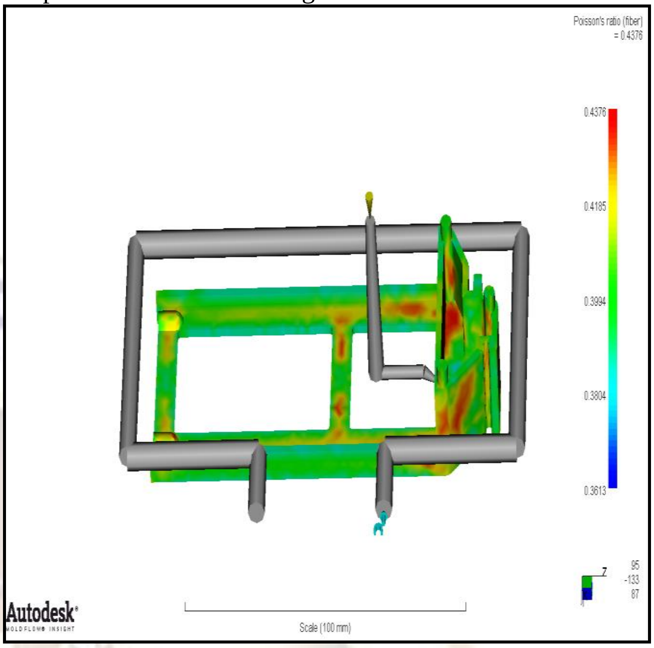

Fig. 6. Poisson's ratio distribution

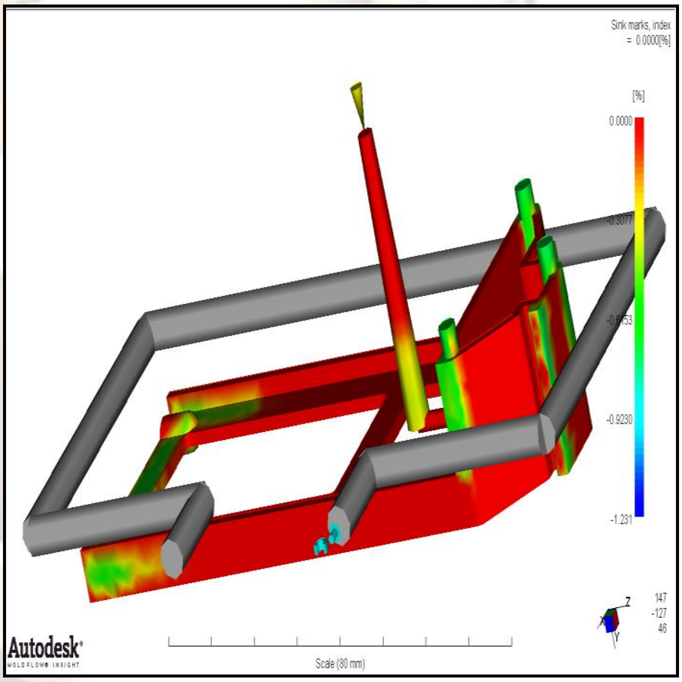

Fig. 7. Location and distribution of Sink marks in the part.

3.1.6. Air Traps Result

The Air traps result is shown in figure 8 shows 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 locations so that the air traps form in easy-to-vent areas, such as at the parting plane.

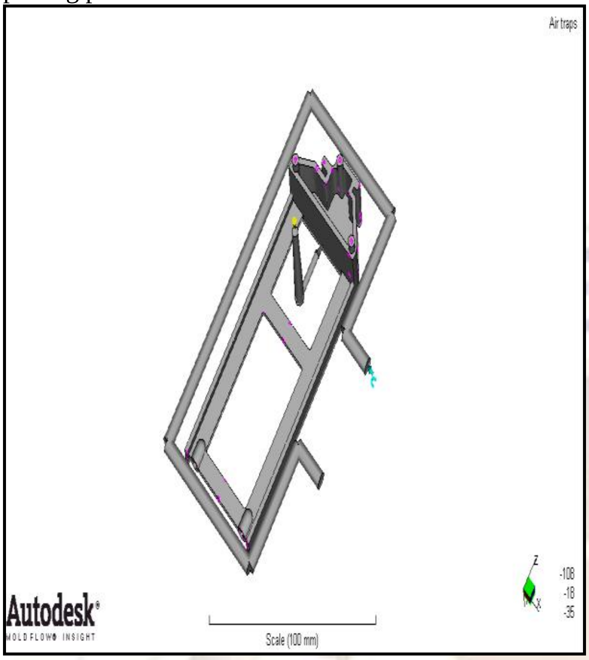

Fig.8. Distribution of Air traps in the melt.

3.2. Pack Analysis

The results generated in pack analysis highlights on the following

1. In cavity residual stress in first principal direction, which shows the stresses in orientation direction of the ejection
2. In cavity residual stresses in second principal direction which is perpendicular first principal direction.

3.3. Warp analysis

This warp analysis results show the deflection at each node of the component along x, y and z axis. This highlights the effects of differential shrinkage, deflection at each node, effect of differential cooling and corner effects on the deflection are shown in figure 9.

3.4. Cooling Analysis

This cooling analysis results help heat removal efficiency as shown in figure 10.

After the analysis the manufacturing of the mold was carried out as per the drawing specification using conventional and non conventional machining process. After the tool is manufactured and assembled the tool is tried out on injection molding machine to see that component made is true to the geometry and dimension specified by the customer. During the trail the problems occurred and the actions taken are listed below.

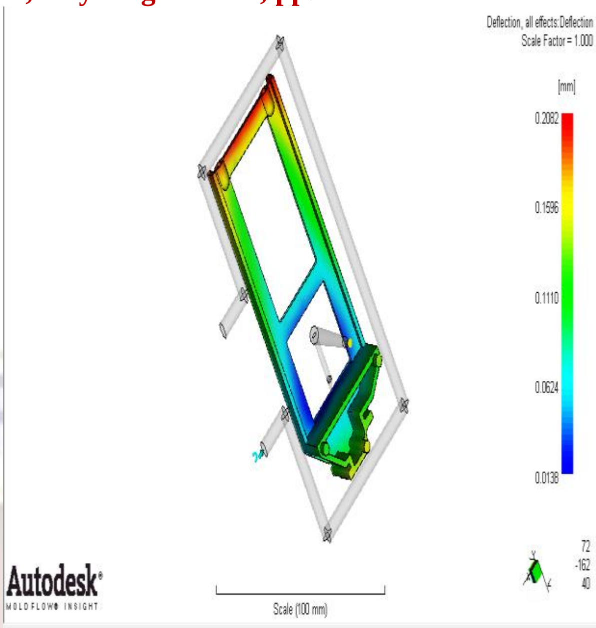

Fig.9. Warp analysis results plot

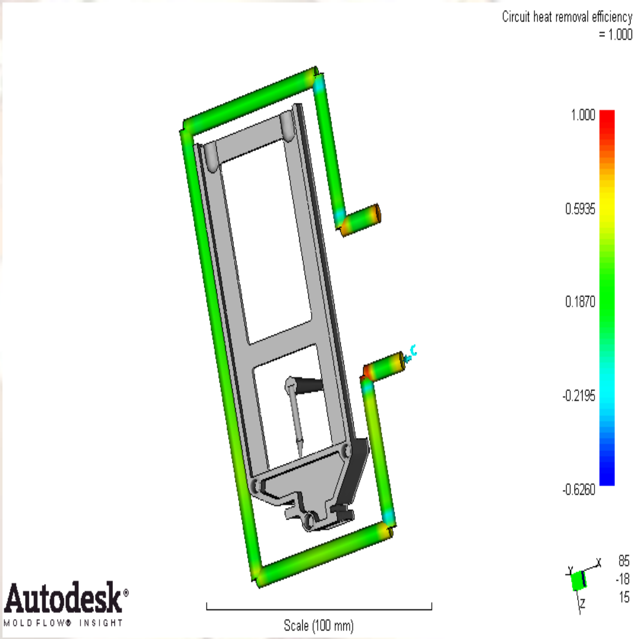

Fig. 10. Cooling analysis plot

Table 1. Problems occurred during tryout.

Sl. No.	Problems	Actions Taken
1.	Flash occurred around the components	Reduction in material temperature, maintained constant cycle time, reduced screw speed, polishing of core and cavity inserts and side cavity.
2.	Ejector pin mark depth is more	Check the ejector pin level and match both half of the tool and excess pin height is ground.

Finally after rectification the component produced are as per customer's specification.

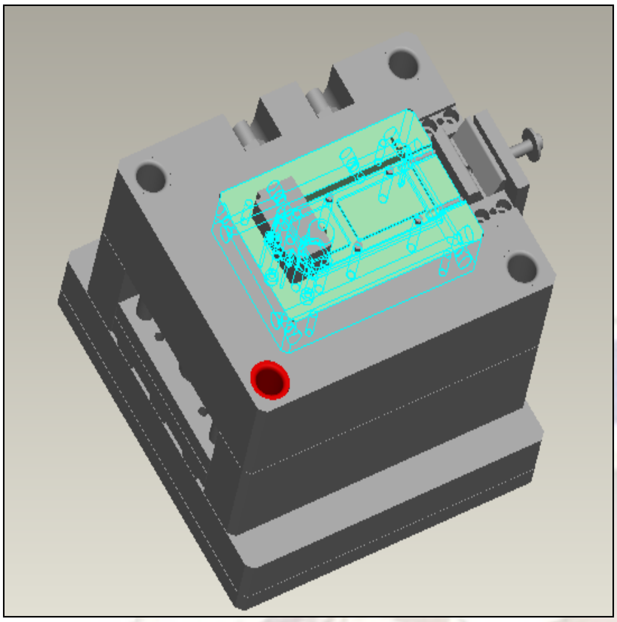

Fig.11. Mold exploded view-Moving Half

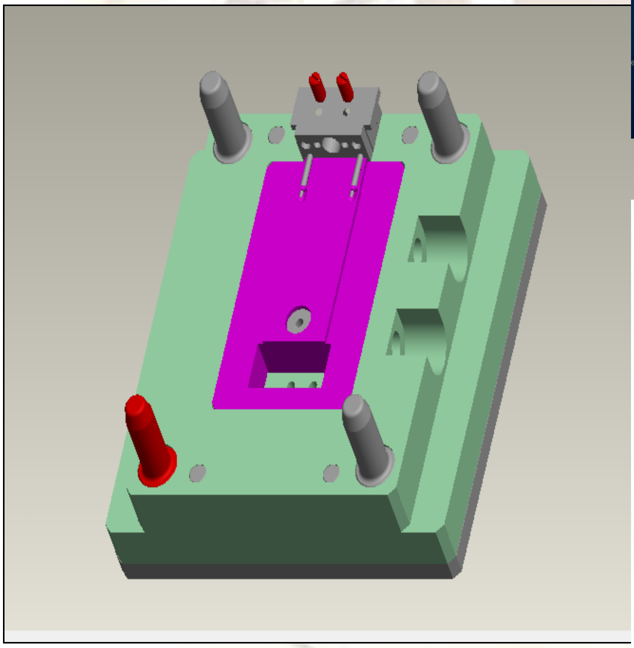

Fig.12. Mold exploded view -Fixed Half.

IV. Poly-propylene with 20% Glass Fiber- Property Details

Component Material tried out - Poly-propylene with 20% Glass fiber

Physical Properties [11]:
Density: 0.914 gm/cc.
Mould Shrinkage: 0.01 - 0.0302 mm/mm.
Water absorption: 0.01 - 0.05%.
Melt Flow rate: 2-34 gm/min.

Mechanical Properties:
Tensile strength: 24-100N/mm^2
Tensile modulus 1380-4520 N/mm^2
Flexural Modulus 1280-5520 N/mm^2
Impact strength: 37 N/mm^2

Thermal properties:
Melting temperature: 200^{0}C - 225^{0}C
Thermal conductivity 0.288-1.2 W/mK
Specific heat capacity: 0.46 J/KgK

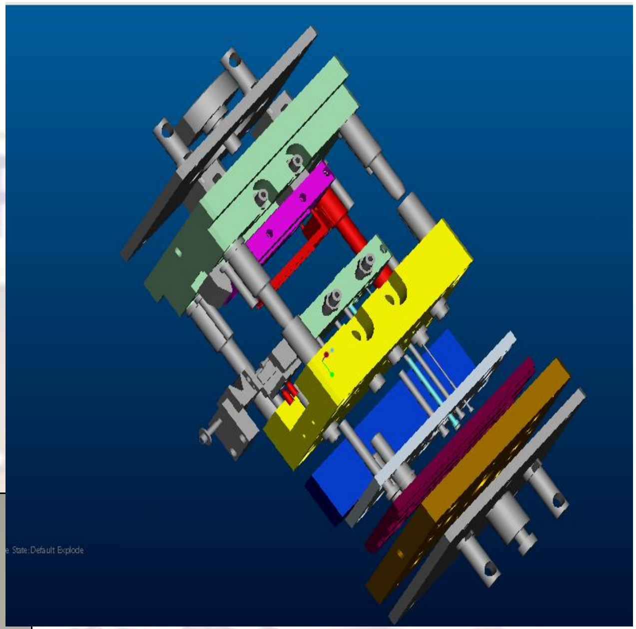

Fig.13. Exploded view of Die Tool

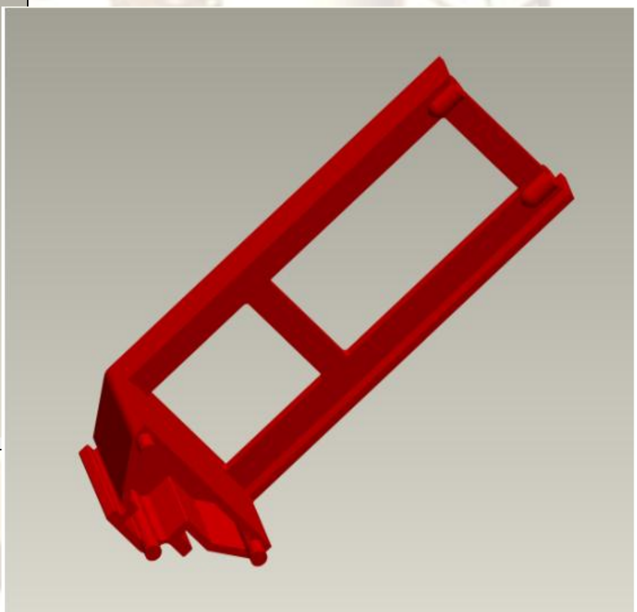

Fig.14. FRP (RAM) Component used in Electrical Transformer

The following problems were noticed during poly propylene with 20% glass fiber components produced by injection molding processes. Their causes and possible solutions are also suggested in the following table 2.

V. Conclusion

In this paper, the design and analysis of injection molding tool for a given FRP component is given in a step by step way along with a flow chart to be followed in methodical way. The optimized design concepts are discussed and selected for the final design and the same was developed. The tool design process involved in the design of various injection molding elements such as cavity and core insert, housing plates, side core actuation, ejector plates, top and bottom plates, etc. After designing the above elements, the various type of analysis were carried out on the tool. Mold flow analysis was carried out on the component and feed system of injection molding tool. This gave satisfactory results and the same was confirmed from analysis such as injection pressure, fill time, flow front temperature, quality of fill, weld line, air traps etc. The results indicated that the injection molded components could be manufactured with minimum molding defects. The tool was manufactured using the CNC and NC machining process according to DME standard and manufactured elements were assembled. The trial out of the injection mold tool revealed the components produced without defective. Further work can be carried out by performing the stress analysis to core and cavity inserts using ANSYS software for more effective design. Fatigue analysis can also carried out for the tool which results in improving the life of the tool. The mold flow analysis can further be used to carry out for design of experiments for fill analysis and pack analysis and the result can be utilized for further optimization of the tool design.

REFERENCES

[1] Haddout, A. and Villoutreix, G., An experimental study of fiber orientation in injection moulded short glass fiber reinforced polypropylene/ polyarylamide composites (Conservatoire National des Artset M6tiers, France), pp.147-153, 1992.

[2] J.A. Brydson, Plastic materials, British Cataloging sixth edition, 1995, Butterworth Heinemann Ltd.

[3] Bown, J, Injection Moulding of Plastic Components, McGraw-Hill, 1979.

[4] Dym, J.B, Injection Moulds and Moulding, Van Nostrand Rheinhold, 1979.

[5] RJ Crawford, Plastic Engineering, Maxwell Macmillan International edition Publication.

[6] Technical paper by Johanna Lamminmäki, Totti Lindgren, Jukka Silén and Heli Vesanto, Influence of injection molding process parameters on mechanical properties of polypropylene-specimen.

[7] Dominick V. Rosato and Donald V. Rosato, Injection Molding Handbook, 1st Edition, 1993, Hanser Publishers.

[8] Injection molding processing and selection of injection molding machine guide, http://www.honeywell plastics.com.

[9] Plastics Materials and Processing Second edition by A. Brent Strong, Bringham Young University.

[10] Plastic industry injection molding applicable note, http://www.deltacompsys.com.

[11] Nello Pasquini, Polypropylene hand book, http://www.hanser.de/ 3 -446-22978-7.

[12] Polypropylene Molding Information obtained by sales@3d-cam.com.

[13] R.G.W.Pye, Injection Mould Design, 4th Edition, 1989, Longman Scientific and Technical Publications.

[14] Mould design, http://www.dow.com/styron/design /guide.

[15] Design Solutions Guide, http://www.honeywell plastics.com/

[16] Villalon, ADA VENUS, Electron Beam Fabrication of Injection Mold Tooling with Conformal Cooling Channels.

[17] Glanvill & Denton “Injection mould design fundamentals” 1st Edition 1965, The Machinery Publishing Co. Ltd.

[18] Steel for Molds English .pdf from Assab steels, www.assabsteels.com.

[19] Book J. Harry Don Bose and Mayne I pribble, Plastic Mould Engineering Hand, Van Nostrand Reinhold Company Publication.

[20] Technical Directory on design and tooling for plastics, CIPET, Guindy, Chennai.

[21] Data Hand Book on plastics, CIPET, Guindy, Chennai.

[22] Banga and Sharma Economics and Cost Estimation.

[23] Injection molding problems and solutions, http://www.exxonmobilpp.com.

Table 2. Causes and Possible solutions during injection molding processes [23]

S.No.	Problems	Causes	Possible Solutions
1	Sink Marks	Part is underfilled or has excessive shrinkage in thicker sections	• Increase shot size • Maintain adequate cushion • Increase cavity or hold pressure • Increase hold time • Reduce fill rate • Cool sink area faster • Open gates.
2	Shrinkage	Volume decreases as Plastic cools and crystallizes or part is not fully packed out due to gates freezing off too soon or insufficient cooling time	• Excessive shrinkage – Increase cavity pressure and hold time • Part oversized or not enough shrinkage – Decrease cavity Pressure • Maintain adequate cushion • Increase hold time.
3	Flash	Insufficient clamp force, mold surface is deflecting, mold shutoff surfaces not seating properly.	• Decrease peak cavity pressure (decrease fill rate and/or use profile injection) • Decrease melt temperature • Increase clamp force • Clean mold surfaces • Check mold surface for flatness • Change gate location.
4	Warp	Non-uniform stress due to excessive orientation and/or shrinkage	• Part ejected too hot (increase cycle time) • Mold at high temperatures, low pressures, and moderate fill rates • Decrease injection fill rate • Improperly balanced core and cavity temperature • Minimize hot spots in mold • Flow too long, insufficient gates • Change gate location
5	Poor Appearance	Flow front slips-sticks on mold surface, jets, or pulsates	• Increase cavity pressure • Fill speed and/or packing time too low • Increase melt and/or mold temperature • Cool more slowly • Mold temperature non-uniform or too low • Increase venting • Improper gate location or design.
6	Sticking in Mold	Over packing, excessive shrinkage, tool design causes physical attachment to the core or cavity	• Over packing, injection pressure too high – reduce • Under packing, excessive shrinkage – see solutions to Short Shot • Increase cycle time (sticking in cavities) • Insufficient knockouts • Remove undercuts • Increase draft angles
7	Gate Blush, Delamination or Cracking at the Gate	Melt fracture	• Adjust injection speed (increase or decrease) • Modify gate geometry (e.g. gate too small, land too long) • Add cold slug wells in runners • Increase melt and/or mold temperature
8	Short Shot	Underfilled part	• Increase shot size • Increase fill speed, pack pressure, and/or injection time • Increase melt and/or mold temperature • Inadequate melt flow rate (use higher MFR material) • Undersized gates, runners, and vents