Optimizing the Die Design Parameters of a Two Cavity Injection Moulding Tool for a Fan Blade Back Cover using Mold Flow Analysis

Dr. Mohammed Yunus[1], Dr. Mohammad S. Alsoufi[2], Mohammed Salman Mustafa[3]

1. Department of Mechanical Engineering, Umm Al-Qura University, Makkah, Kingdom of Saudi Arabia
2. Department of Mechanical Engineering Umm Al-Qura University, Makkah, Kingdom of Saudi Arabia
3. Department of Mechanical Engineering, KNS Institute of Technology, Bangalore, India.

ABSTRACT

For producing intricate and complex shape plastic parts with higher dimensional accuracy, injection molding method is widely used. Presently, plastic industries are under great pressure due to the globalization of the market, the short life cycle of the product, increasing diversity, the high demand of product quality, meeting   requirements by adapting various advanced technologies like CAD/CAM/CAE, concurrent engineering and so on for the development of the injection molded part. The design of a mold is of high priority for product quality and economic processing and also it acts as an integral part of plastic injection molding since the quality of final plastic part is greatly reliant on injection mold. A two cavity injection molding tool design and its mold flow analysis for a given component was taken up according to the industrial specification. Polypropylene (PP) is thermoplastic material used for Fan Blade Back Cover due to its light-weight, durability, recyclable, rugged and resistant to chemical solvents. The design of tool includes the design of core insert, cavity inserts, number of cavities in the mold, type of mold, feeding system, side cores and production quantity. After thorough study and necessary calculations of the component for achieving 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. 3D virtual models are created using the Solidworks software. The 2D drawings were drafted using Auto-CAD software. The core and cavities are extracted through modeling software and Mold Flow Analysis was carried out. Complete mold design is designed after conducting the mold flow analysis and optimizing the gating and the runner design by CAE software for flow and solidification of the liquid material into the cavities. It determines the optimal combination of part geometry, material choice and processing parameters to produce quality finished parts. This work also reveals results of fill analysis, pack analysis, warp analysis and cooling analysis. Finally, based on the analysis results, the manufactured mold produced was tested for the production of final product, with precise specification of the industry requirement. The present work can also be further extended for a detailed stress and fatigue analysis for the core and cavity inserts to obtain improved tool life.

Keywords - Injection Molding, Polypropylene, Mold Die Designing Methodology, Mold Flow Analysis, CAE analysis.

1.1.   INTRODUCTION

Injection molding is a cyclic process of forming plastic into a desired shape by forcing the material under pressure into a cavity. It is one of the most common and versatile operations for mass production of complex plastics parts with excellent dimensional tolerance. It requires minimal or no finishing or assembly operations. In addition to thermoplastics and thermosets, the process is being extended to such materials as fibers, ceramics and powdered metals, with polymers as binders. PP material for the component and the screw type injection-moulding machine are the preferred choices. The architecture of an injection mould design system is proposed based on the practical mould design procedures. Mold and die making is an important industry since their related

products represent more than 70% of the non-standard components in consumer products. Therefore the combined effect of minimized surface area of undercut, flatness of parting surface and minimized draw depth is considered for the determination of best parting direction, parting line and surface for a two-piece permanent molded component. The mold flow analysis for optimal gate design of thin-walled injection molding show that the gate for single point of two sides is suitable on thin-walled injection molding by numerical simulation. In plastic injection molding, they are molded by core, cavity or side-cores.

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 is used to turn plastic stock into finished products. The process involves many steps [14]:

- Feeding raw material
- Plasticize the raw material
- Fill the mould
- Pack the mould
- Hold pressure
- Cooling of mould
- Opening of mould and Part ejection

The critical items to be considered while designing a injection mould tool are [3,4]

- Shrinkage.
- Selection of Parting Surface.
- Number of Cavities.
- Feed System.
- Cooling System.
- Ejection System.
- Venting.

The aim of this work is to develop an intelligent CAD tool that supports EDM electrode design. The present work covers the Design, Analysis and Manufacturing of two cavity injection mold for fan blade back cover of polypropelene. The design of the tool is aimed for a high productivity and to obtain good quality of products economically.

2.   METHODOLOGY

Methodology is a systematic approach for the realization of total task. It consists of the following details:

♦   Study of Component drawing.
♦   Component Description.
♦   Plastic flow Analysis.
♦   Solid modeling of the tool.
♦   Selection of tooling materials.
♦   Tool manufacturing, assembly and tryout.

2.1.   Study of Component Drawing: This is most important and the first step for the designer. The component drawings are carefully scrutinized to extract the maximum possible amount of information available such as critical dimensions, line of draw, parting line, suitable ejection system, and required side core.

2.2.   Component Description: The component is a back cover of fan blade unit which plays an important role in the assembly of the complete equipment. The component is designed in such a way that it conceals the complete assembly and also balances outside with other adjoining parts or

assemblies due to which the cover is designed in such a way as to ensure both the applications. There is a hole in the side surface of the component to facilitate the necessary cables for the fan motor after assembly. As per requirement a two cavity injection moulding tool is to be developed [21].

Feeding system design is simple. Conical sprue is connected along with the modified trapezoidal runners. Runner balancing is done such a way that all two cavities are filled simultaneously. Hence component shape is small and circular fan gate is used. Coming to the ejection system pin ejection system is used. Six pin ejectors are used for feeding system ejection
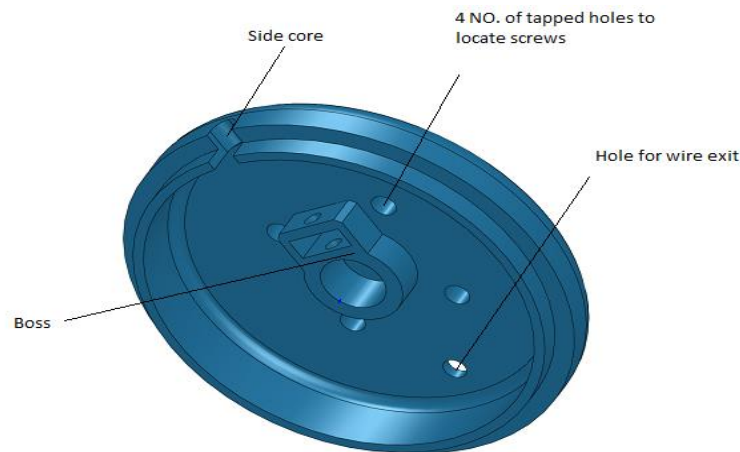
Figure 1. 3D View of a Fan blade back cover

2.3.   Plastic Flow Analysis: Plastic flow analysis is done using Autodesk software, to know the optimum gate location, temperature distribution, fill time, air traps, weld lines etc. and also tool modeling environment used is solidworks software[22].

2.4.   Solid Modeling of the Tool: Solid modeling of the tool is done using high end CAD solidworks software. The component weight, volume and surface area, projected area, etc. were determined using the same software package. This data is helpful to do design calculation. 2D – drawings of the mold parts are prepared for manufacturing purpose.

2.5.   Selection of the Tooling Material: Suitable material is chosen for manufacturing of each part of the tool depending on their primary function and use. The different tooling materials used for each part of the tool is given in bill of materials.

2.6.   Tool Manufacturing, Assembly and Tryout: Tool manufacturing has been done by referring 2D-drawings followed by the use of different manufacturing techniques stage by stage. After manufacturing all the tool parts the blue matching of mating parts of core and cavity has been done and the final assembling has been done. The final stage of molding is tryout stage, the tool is loaded on the injection molding machine and then samples are taken out injection molding process.[11,12]

3.   MOLD FLOW ANALYSIS

The mold flow software is designed to determine the manufacturability of the plastic part design using model of a component drawn in a CAD package and it is an important analytical tool available for various analysis of injection molded components. The sequence of work involved in mold flow analysis is given under.
1. Converting the 3D model in STEP format.
2. Importing the converted file to the solver package specifying the boundary condition, loads such as injection pressure, injection time, mold temperature, material properties etc.

3. Build the feed system such as sprue, runner and gate, cooling lines.

4. Mesh the feed system and cooling lines.

5. Run the analysis for different analysis types like fill, flow, cooling, warpage etc.

6. Study the result, interpret them.

7. Establish the optimized data for runner, gate, sprue dimensions, coolant temperature etc.

8. Incorporate this value in the design; send the optimized process setting to molding shop for trial.

When CAE is used early in the design stage the cost saving are substantial not only because of time saving, but also the avoidance of expensive and common last minute mold modifications.

After the analysis we will have the following results.

1.   Fill time.

2.   Frozen layer fraction at end of fill result.

3.   Pressure result.

4.   Pressure at injection location result.

5.   Pressure at velocity/pressure switchover result.

6.   The weld lines and

7.   Cooling analysis

8.   Sink, index result.

Table 1. Processing parameters

Injection pressure	150 MPa
Recommended Melt Temperature	200°C
Recommended Minimum Mold Temperature	65°C

4.   RESULTS AND DISCUSSION

4.1. 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 as shown if figure 2. In a part with a good fill time result, the flow pattern is balanced. In a balanced fill time results we can see the following:

•   All flow paths finish at the same time. The flow fronts should reach the extremities of the model at the same time. This means that each flow path should end with red contours.

•   The contours are evenly spaced. The contour spacing indicates the speed at which the polymer is flowing. Widely-spaced contours indicate rapid flow, while narrow contours indicate that the part is filling slowly.
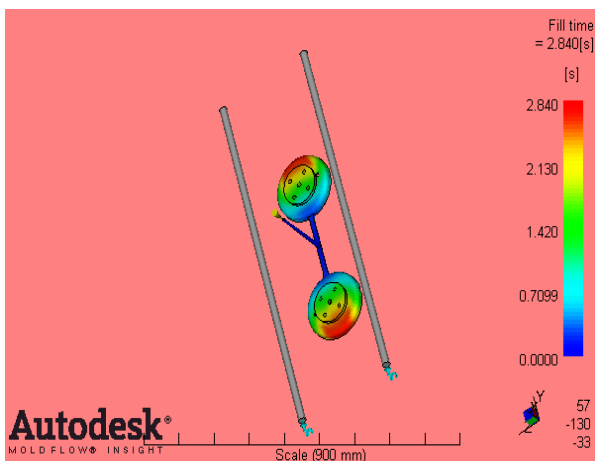

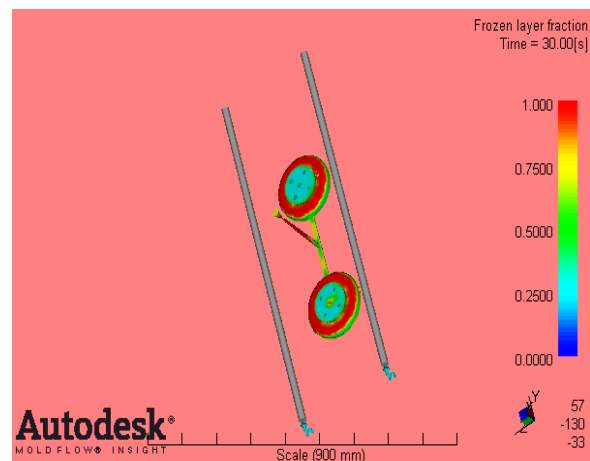Figure 2: Fill Time.   Figure 3: Frozen layer fraction at end of fill.

Short shot: On the Fill time result, a short shot will appear as translucent. Check the ends of the flow paths for any translucent areas. For models analysed using 3D analysis technology, you can also use the unfilled cavity result to see if there are any unfilled sections in the interior of the part.

Over packing: If a Fill time result shows that one flow path finishes before others do, it may indicate overpacking and can cause high part weight, warpage and non-uniform density distribution throughout the part.

Weld lines: Overlay the Weld line result on the Fill time result to confirm their presence. Weld lines can cause structural and visual defects.

Air traps: Overlay the Air trap result on the Fill time result to confirm their presence. Air traps can cause structural and visual defects.

4.2. Frozen Layer Fraction at End of Fill

The Frozen layer fraction at end of fill result represents the thickness fraction of the frozen layer at the end of filling as shown in figure 3. It ranges from 0.0 to 1.0. A higher value indicates a thicker frozen layer (or thinner flow layer) and higher flow resistance. A polymer is considered frozen when the temperature falls below the transition temperature. Frozen-layer fraction has very significant effects on the flow resistance. The viscosity exponentially increases with decreasing temperature. The thickness of the flow layer is also reduced as the thickness of the frozen layer increases. The effect of the thickness reduction can be roughly estimated with the definition of fluidity, as with representative shear rate. The fluidity is proportional to the cubic power of the part thickness. A 50% reduction in part thickness reduces the fluidity by a factor of eight (or increases the flow resistance by a factor of eight). Moreover, a 50% reduction in thickness in runners reduces the fluidity by a factor of 16. It is not surprising that excessive high pressure is required to fill parts in which hesitation occurs early in the filling stage. The flow layer becomes very thin in areas of hesitation, which are filled last. The frozen layer fraction generally will be very low near the injection location and the end of fill. The maximum frozen layer fraction at the end of fill should be less than 0.20–0.25. Higher values will make the part difficult to pack out. Areas of the mold that are filled early in the cycle, but have little subsequent flow, normally have the highest frozen layer fraction. None of the part should have a frozen layer fraction higher than 0.20–0.25 at the end of fill. A faster injection time will reduce the frozen layer fraction. Time to reach ejection temperature result. If hot spots are occurring, look for ways to change the cooling circuit design.

4.3. Pressure

The Pressure result is generated from a fill analysis shows the pressure distribution through the flow path inside the mold as indicated in figure 4. 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 magnitude of the pressure (or pressure gradient) depends on the resistance of the polymer in the mold, because polymer with high viscosity requires more pressure to fill the cavity. Restricted areas in the mold, such as thin sections or small runners, and long flow lengths also require a larger pressure gradient and thus higher pressure to fill. Normally the maximum injection pressure at the nozzle is about 140 MPa (20,000 psi). We recommend having a maximum pressure of 100 MPa (14,500 psi) for the mold (part and feed system) and 70 MPa (10,000 psi) maximum for the part. There are many molding machines with higher pressure capacities. The filling phase should be controlled by velocity, not pressure. Pressure should be zero at the extremities of each flow path at the end of filling. Generally, the maximum pressure should be less than 100 MPa for parts with runners and 70 MPa for parts without runners.
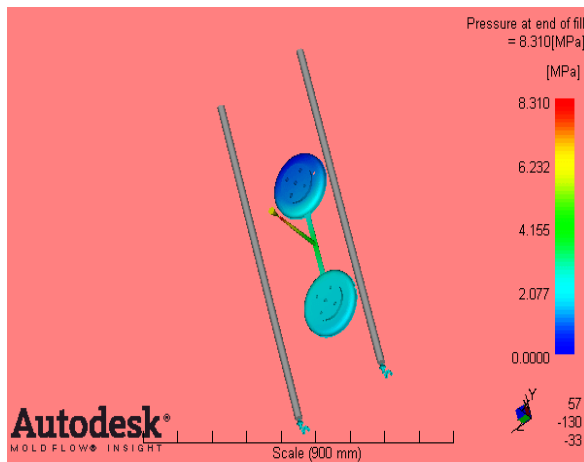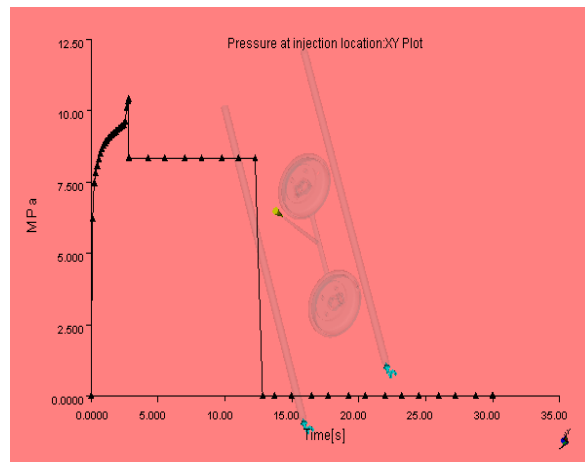
Figure 4: Injection Pressure.   Figure 5: Pressure at injection location: XY Plot

4.4. Pressure at Injection Location

The Pressure at injection location result shows the pressure at the injection location at various times during the filling and packing phases of the analysis as shown in figure 5. All of the injection locations on the model will have the same pressure. This is an assumption used in the analysis. The Pressure at injection location result is very useful for checking whether there are any pressure spikes, which is normally a sign of imbalance. This could be within a part or between parts. If it is within a part, normally this could be fixed by changing gate location(s). Sometimes only a subtle change is necessary. The Pressure at injection location result is an XY plot generated from Fill and Microchip Encapsulation analyses. A significant change in the slope of the pressure curve, during the filling phase, indicates a change in the pressure gradient in the part. Often this is caused by a flow imbalance within a part or for multi-cavity models between parts. Review the fill time result and other pressure results for balance problems. The packing profile is shown with this result. The default packing profile uses 80% of the pressure at switch over for 10 seconds. On this result, the pressure will drop instantaneously 20% then the line will be horizontal for 10 seconds. This result can also be used to ensure the packing profile is correct.

4.5. Pressure at Velocity/Pressure Switchover

The Pressure at V/P switchover result is generated from a Fill analysis, and shows the pressure distribution through the flow path inside the mold at the switchover point from velocity to pressure control. From figure 6, pressure should be zero at the extremities of each flow path at the end of filling. During the filling stage, large variations in the pressure distribution, indicated by closely-spaced contours, should be avoided. Normally the maximum injection pressure at the nozzle is about 140 MPa (20,000 psi). We recommend having a maximum pressure of 100 MPa (14,500 psi) for the mold (part and feed system) and 70 MPa (10,000 psi) maximum for the part. There are many molding machines with higher pressure capacities. If pressure capacity is not known, assume it has 140 MPa. If the pressure capacity of the molding machine is known, use no more than about 75% of the pressure capacity for a design guide for the entire mold, and 50% for just the part. There will be zero pressure at the flow front, and some portion of the part is not filled. The portion not filled as shown in this plot will be filled under pressure control as defined in the process settings.
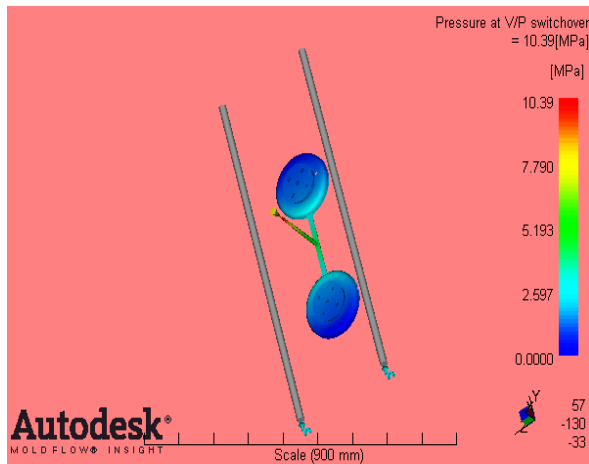
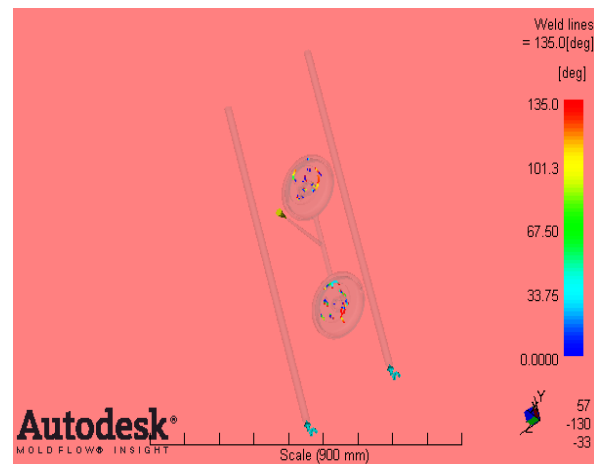Figure 6: Pressure at V/P Switchover.   Figure 7: Weld lines

4.6. Weld Lines

The Weld Lines result in figure 7 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.

4.7. Cooling Analysis

This topic specifies the results generated by a Cool analysis using a thermoplastic material without fiber or filler content. The Cool analysis contains the change in coolant temperature from coolant-in to coolant-out.

4.7.1. Circuit heat removal efficiency

The Circuit heat removal efficiency result provides a measure of the effectiveness of each cooling channel section in extracting heat from the mold during the molding cycle.This quantity shows the relative efficiencies within the cooling system. This is shown in figure 8. In most cases, the cooling channels are cooling the mold, and the plot values are positive. The section with the highest efficiency is assigned a value of 1. All other heat removal   efficiencies are represented as a fraction less than 1. In cases where the cooling channels are heating the mold, the plot values are negative. The section providing the most heat input is assigned a value of -1, and all other heat input efficiencies are represented as a fraction greater than -1.
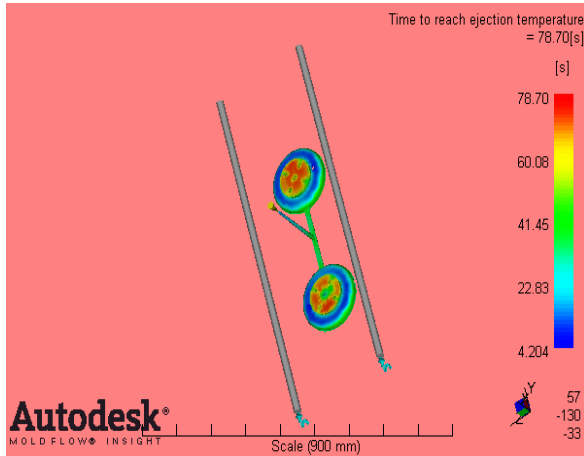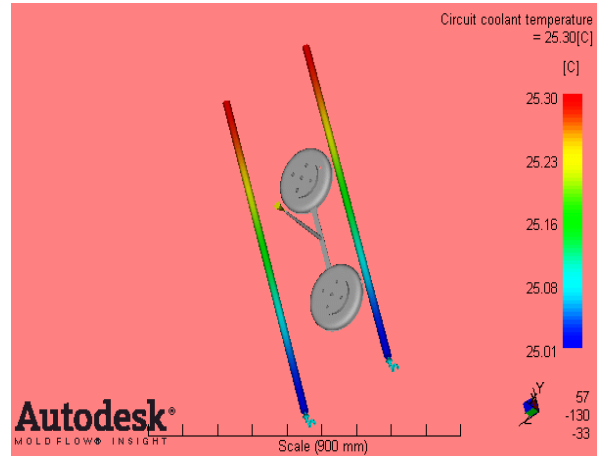
Figure 8: Circuit heat removal efficiency   Figure 9: Circuit coolant temperature

4.7.2. Circuit coolant temperature

The Circuit coolant temperature result shows the temperature of the coolant inside the cooling circuit(s) as shown in figure 9. The Cool analysis log contains the change in coolant temperature from coolant-in to coolant-out. If the increase is unacceptable, (greater than 2–3°C), use the circuit coolant temperature result to determine where the greatest increase in temperature occurs. In parallel circuits, the final increase in coolant temperature from coolant-in to coolant-out may be small. However, the coolant may have reached a high temperature in some sections of the cooling lines. The outcome of this analysis when coolant flows through one line shows that temperature of the coolant increases mixes with low temperature coolant and coolant leaves the circuit.

4.7.3. Circuit flow rate

The Circuit flow rate result shows the flow rate of the coolant inside the cooling circuit(s).This is shown in figure 10. Use this result if you have set a minimum Reynolds number. This result is to determine if the flow rate required achieving turbulent coolant flow.
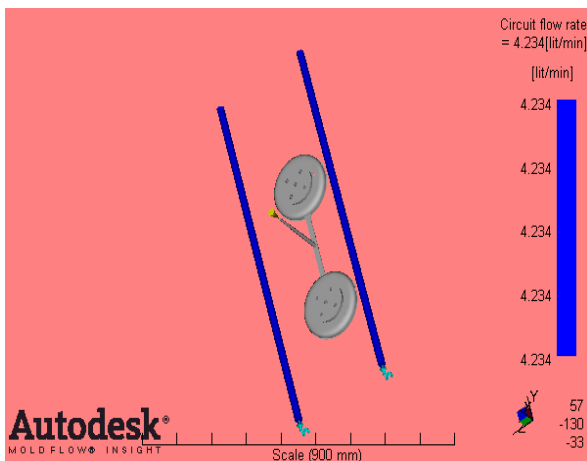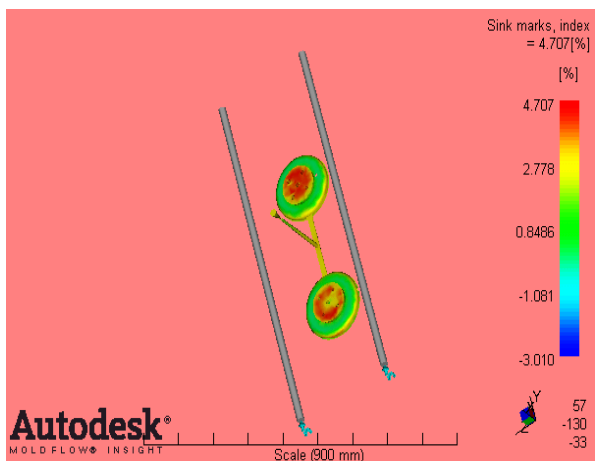

Figure 10: Circuit flow rate   Figure 11: Sink marks

4.7.4. Sink marks, index

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. The Sink marks, index generated indicates the likely presence and location of sink marks (and voids) in the part. This is shown in Figure11. The Sink marks, index indicates a degree of severity of depth as affected by the material, part geometry, position relative to the injection location and mold filling conditions. Changing any of these would allow you to determine its contribution to the sink marks severity. Generally, if the thickness of a rib is less than or equal to 60% of the main wall section, then there are likely to be no significant sink marks. The parameters observed from mold flow analysis are shown in the following table 2.

Table 2. Design parameters observed at different locations in mold flow analysis

	At Sprue	At runner	At gate	At cavity
Fill time(Sec)	0.032	0.709	1.420	2.840
Pressure(Mpa)	8.310	4.155	2.077	1.091
Weld lines(deg)	Red	Yellow	Green	Blue
	135	101.3	67.50	33.75
Coolant temperature(C)	25.30	25.23	25.16	25.08
Sink marks(%)	4.707	2.778	0.8486	-

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

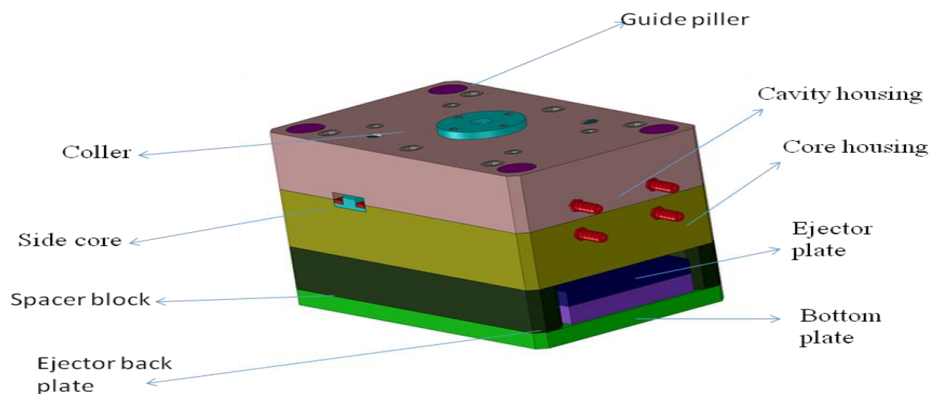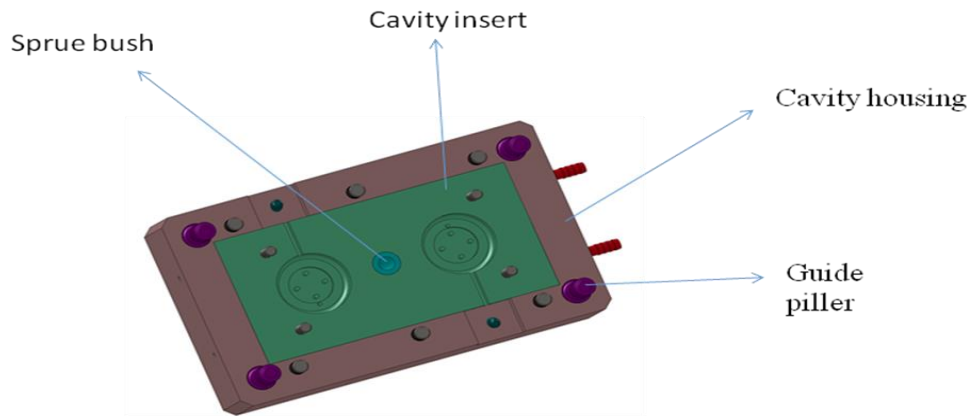**Figure 12: Fixed Half of the Tool**   **Figure 13: Moving Half of the Tool**
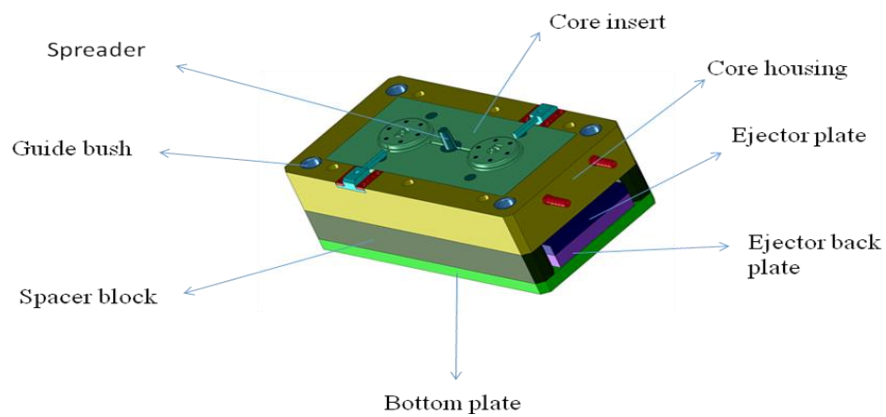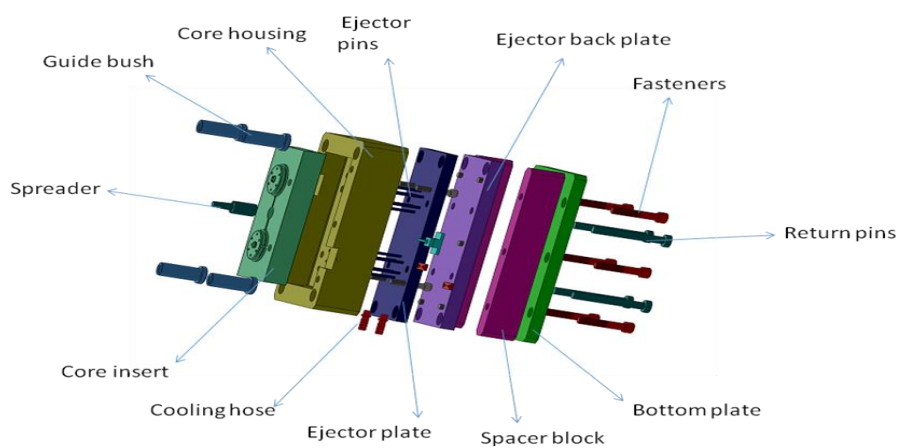

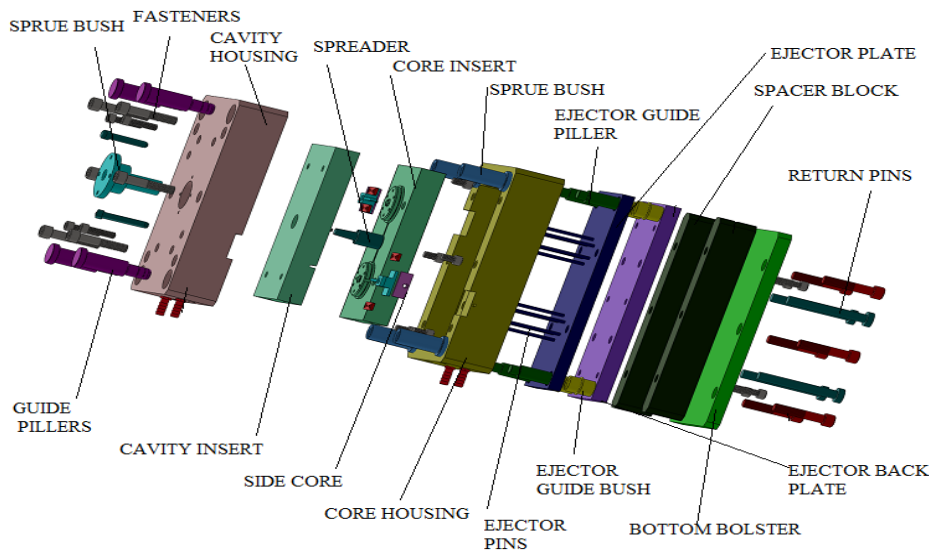**Figure 14: Tool Assembly**

Figure 15: Cavity half

Figure 16: Core Half

Figure 17: Exploded view of Core half

Figure 18: Exploded view of Mould

5. CONCLUSION

In this technical paper, design, analysis and manufacturing of a two cavity injection mould tool of a fan blade back cover is given in a step by step way along with detailed procedure to be followed in methodical way. The optimized design concepts are discussed and selected for the final design of mould tool using CAD softwares like solidworks and Mold flow analysis helps in easy designing of mold and in the optimization of gate and runner system respectively. The Detailed drawing of AutoCAD helps the manufacturing process using CNC machines because here co-ordinate dimensioning method is used and the same was developed. Design of various injection moulding elements such as cavity and core insert, housing plates, side core actuation, ejector plates, top and bottom plates, etc, are the process involved in the design of a tool. After designing the above elements, the various types of analysis were carried out on the tool. Mould flow analysis was carried out on the component and feed system of injection moulding tool. The analysis provided satisfactory results and the same was confirmed from analysis such as fill time, frozen layer fraction at end of fill, injection pressure, pressure at velocity/pressure switchover, weld lines, cooling analysis, sink marks, index, quality of fill, air traps etc. The results obtained shows that the injection moulded components could be manufactured with no moulding defects. Finally, the tool was manufactured using the in-house tool room facilities according to standards specified by the organization. The manufactured mould elements were then assembled. The trial out of the injection mould tool revealed the components produced without defective. The work can be further extended by carrying out the stress analysis and fatigue analysis to core and cavity inserts using standard CAE software for more effective design and life improvement of the tool. The results of mould flow analysis can further be used to carry out for DOE (design of experiments) followed by ANOVA and confirmation test 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]   Dr.J. Fazlur Rahman, Dr.Mohammed Yunus, Mohammed Irfan, Tajuddin Yezdani, T.M., Optimizing the Die Design Parameters for FRP Components Produced in Injection Molding using Mold Flow Analysis, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 2, Issue4, pp.2103-2111, July-August 2012.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

[22]   Data Hand Book on plastics, CIPET, Guindy, Chennai.Banga and Sharma Economics and Cost Estimation.

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