DESIGN OF 'PLASTIC INJECTION MOLD' FOR AN 'AIR VENT BEZEL' THROUGH FLOW ANALYSIS (CAE) FOR DESIGN ENHANCEMENT

Jitendra Dilip Ganeshkar[1], R.B.Patil[2]

[1]*ME CAD CAM Pursuing, Department of mechanical engineering, MGM's Jawaharlal Nehru Engineering College Aurangabad- 431003, Maharashtra, India*

[2]*Associate Professor, Department of mechanical engineering, MGM's Jawaharlal Nehru Engineering College Aurangabad- 431003, Maharashtra, India*

Abstract

Designer's of plastic injection Molds need to study from this point of view, the type of Mold to be designed, the physical Mold orientation, the dimensions for each of the element in the Mold, the location of the gate, runner, requirement of insert, draft angle requirement, shrinkage factor, location of cooling channels. Design can also be simplified by using standardized parts of the mold such as ejector pins. By eliminating the obstruction to smooth flow of plastic a good of quality component can be achieved. The effort of this thesis work is to find out the nuances in the Plastic Injection Mold Design while borrowing the inputs from the Flow Analysis (CAE) conducted for the 'Air vent bezel' to study the behavior of the Melt during flow. The Mold Design would incorporate suitable checking to ensure the best quality product in terms of `defect-free' output. This thesis work consist methodology of plastic injection molding process, material requirement, how to overcome weld mark, shrinkage etc to satisfactorily assemble the component.

Keywords: *Draft angle, Shrinkage, CAE, weld mark*

--***--

1. INTRODUCTION

The cost of the mold is high and any process that is not optimized renders heavy overheads during its development cycle and production. So designing the mold which ensures best suitability for the features on the component with smooth flow of molten plastic is very important part of development process.

The successful launch of any plastic product depends on knowing the true costs and profitability before the job is started. Injection molding typically involves large volumes of parts. Small cost overheads per part can be compounded to large cost differences over the life span of the part. Major cost components considered here are material, re-grind and machine costs. Scrap, rejections and regrind costs are also accounted in the cost.

1.1 Simplified Design

The problems which may occur in the development phase of the product are solved in the design phase itself. So the rework and modification of mold is avoided by minimizing the wastages and scrap.[1]

To insure final quality of a product, it is necessary to start out with quality components. Injection molded parts can be molded to a high quality standard by focusing on these areas of plastic technology:

1) Correct part design
2) Proper selection of material
3) Plastic processing

Product designer may create quality molded parts, which will maximize performance and cost effectiveness, only by using capabilities from above three areas of plastic technology.

2. METHODOLOGY

The following steps are involved for achieving the objectives of the project that can be enumerated as

1. Generation of 3D model of component from 2D drawing.
2. Study the component design with the perspective of a Mold Designer. (Query/ Analysis report)
3. Identification of the critical features that would call for special elements while designing the mold, such as critical dimensions, tolerances, surface finish, abrupt changes in thickness, undercuts.
4. Generation of a rough layout for the mold design. Generate a layout using 3D modeling tools (3D model generation by using tools like CATIA-V5/ UG)
5. Design and detailing for manufacturing the mold. (Final 3D model and 2D drawings)
6. Design a simplified mold as per the functional requirements of the component.
7. Review and get the approval from the product designer for changes in design.
8. Design validation of Mold for cycle time optimization and required level of dimensional accuracy, surface finish and strength.[1]

3. TONNAGE CALCULATION

Clamping Unit of injection molding machine is rated by the maximum amount of clamp force that the machine is capable of producing. This force is required in order to keep the mold closed during the injection process, this is the primary purpose of the clamp unit. The force rating is stated in tons. So, a specific machine having a rating of 200 tons is capable of producing a maximum clamping force equivalent to a total of 200 tons.

3.1 Calculation of Tonnage of Injection molding machine to manufacture Air vent

Polycarbonate & ABS are the material for AIRVENT suggested by sponsoring company. Polycarbonate & ABS are fairly stiff and a lower flow material (more viscous).

The dimensions of Air vent Bezel want to manufacture:

The total clamp force required for a specific product is determined by finding the projected area of that product & projected area is determine by multiplication of length & width of product.[1]

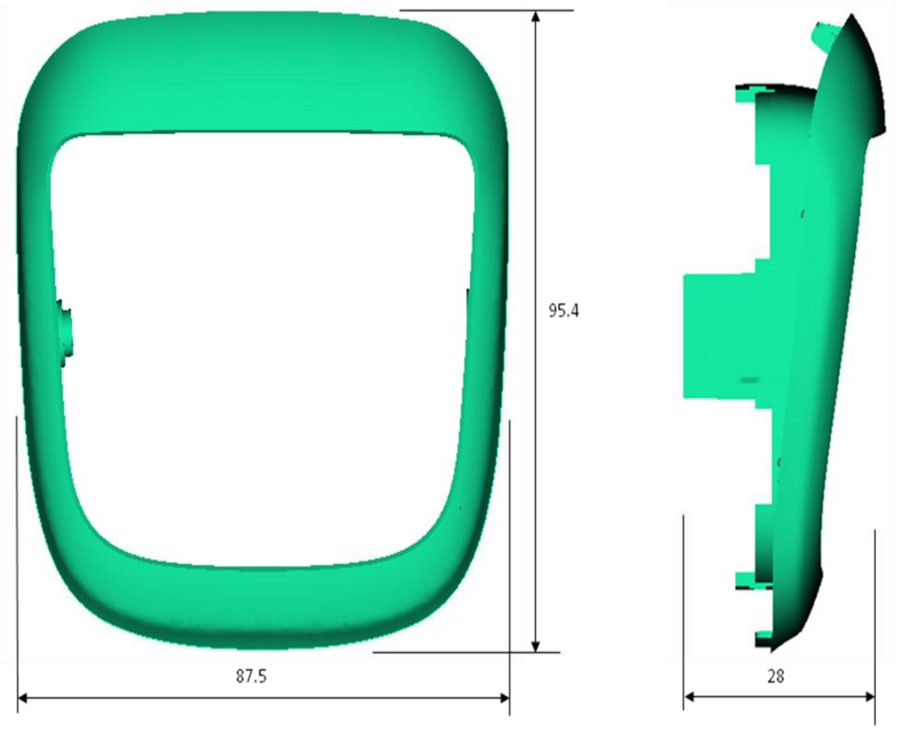

Fig -1: CAD Model of Air vent Bezel

87.5mm length, 95.4mm width, 28mm depth
means 3.44 inch length, 3.75 inch width, 1.10 inch depth

Hence, Projected area = 3.44 x 3.7
= 12.9 $inch^2$

Now the clamp force can be determine by multiplying projected area by clamp factor which is in between 2 to 8 tons per square inch, The lower numbers can be used for high flow materials and the higher numbers can be used for low flow (more viscous) materials. Hence for given material considering clamp factor as 5 based on viscosity of material & on the basis of experience.

Clamp force required = Projected area x Clamp factor
= 12.9 x 5
= 64.5 Tons

As depth of part is more than 1 inch (1.10 inch), hence need to add 10% clamp force & it will get after adding 10% as:
Clamp force required = 64.5+6.45
= 70.95 Tons

Now add 10% for safety factor and the required force increases to:

Clamp force required = 70.95+7.095
= 78.045 Tons

The nearest machine size to that requirement is 100 Tons.

4. INJECTION MOLDING ANALYSIS REPORT FOR AIR VENT BEZEL

Cool, fill, pack and warp analysis is done to study and optimize filling, packing and warpage characteristics of the part with PC+ABS, Bayblend T65 XF, Bayer material science.

4.1 Process Parameters used

Following parameters are used for analysis [5]

Mold temperature used (^{0}C) : 70

Melt temperature used (^{0}C) : 260

Flow rate (cc/s) : 10

V/P Switchover (%) : 99

Pack time (sec) : 5

Cooling time (sec) : 20

Mold Open/Close time (sec) : 5

4.2 Feed System Details

The feed system for feeding the mold is oblong sub gate with sprue is used

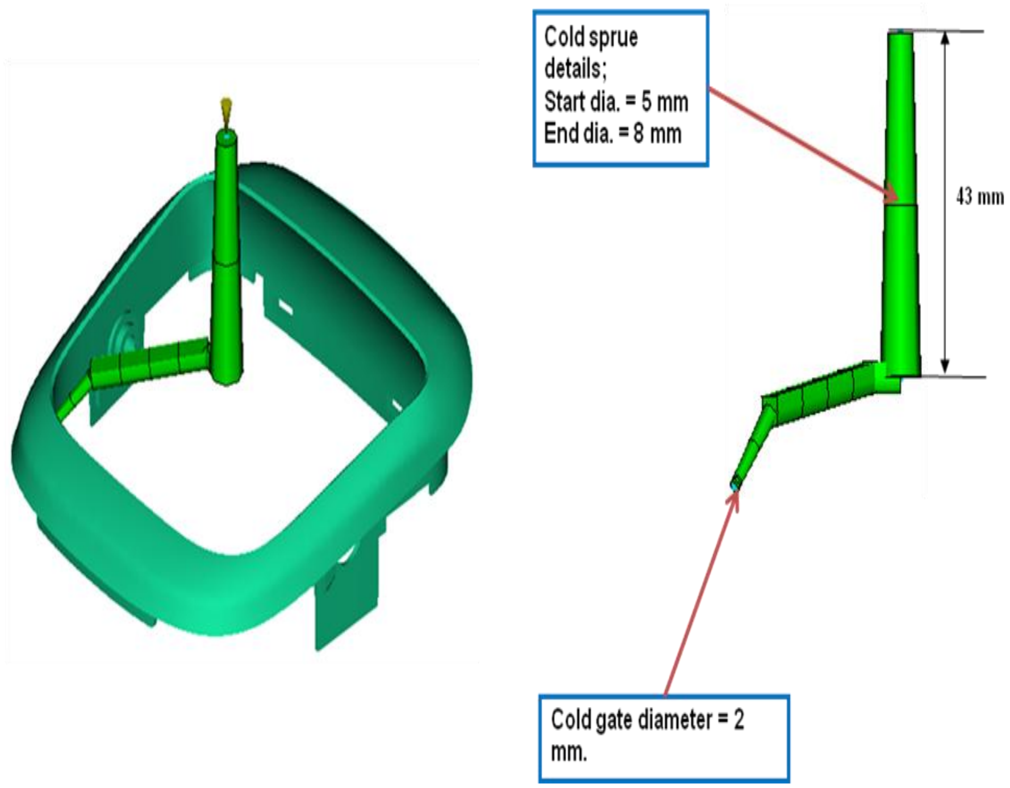

Fig -2: Location of oblong sub gate

4.3 Cool Analysis Results

The circuit for cooling the mold is important to maintain temperature inside the mold. The analysis results of cooling are as follows [2]

Table -1: Cool analysis results

Coolant inlet temp	Coolant outlet temp	Increase in temperature observed
70°C	70.2˚C	0.2°C.

(Increased within recommended temp. of 3°C)

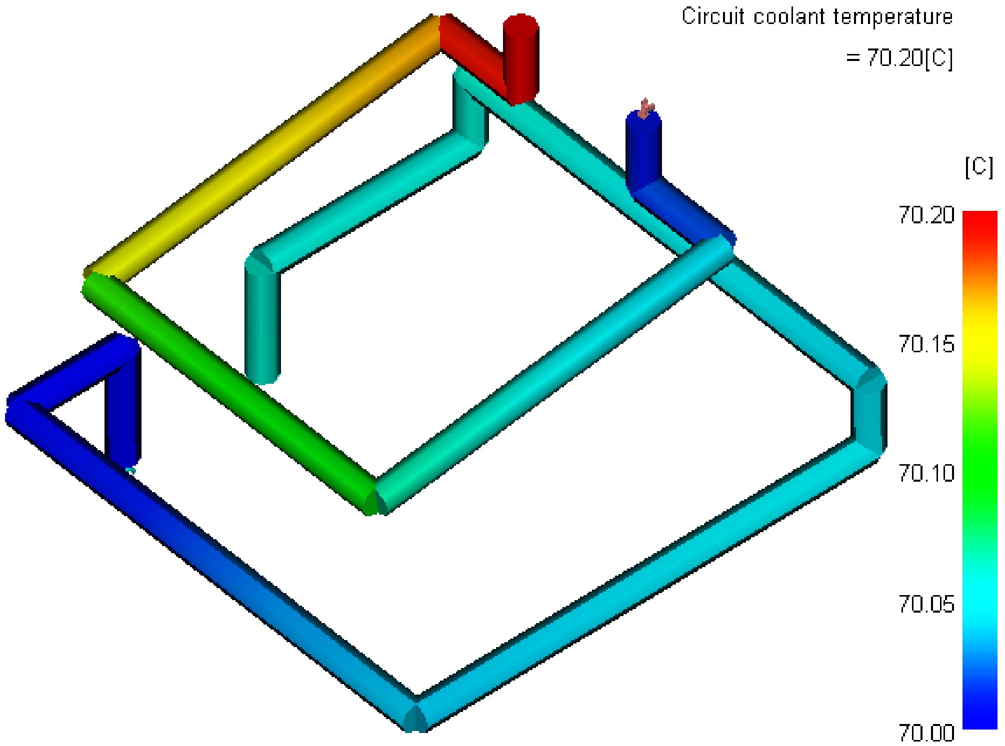

Fig -3: Circuit coolant temperature

4.4 Fill and Pack Analysis Results

The part is filling completely in 1.94s with flow rate 10 cc/s. Following figure shows the part filling at particular intervals

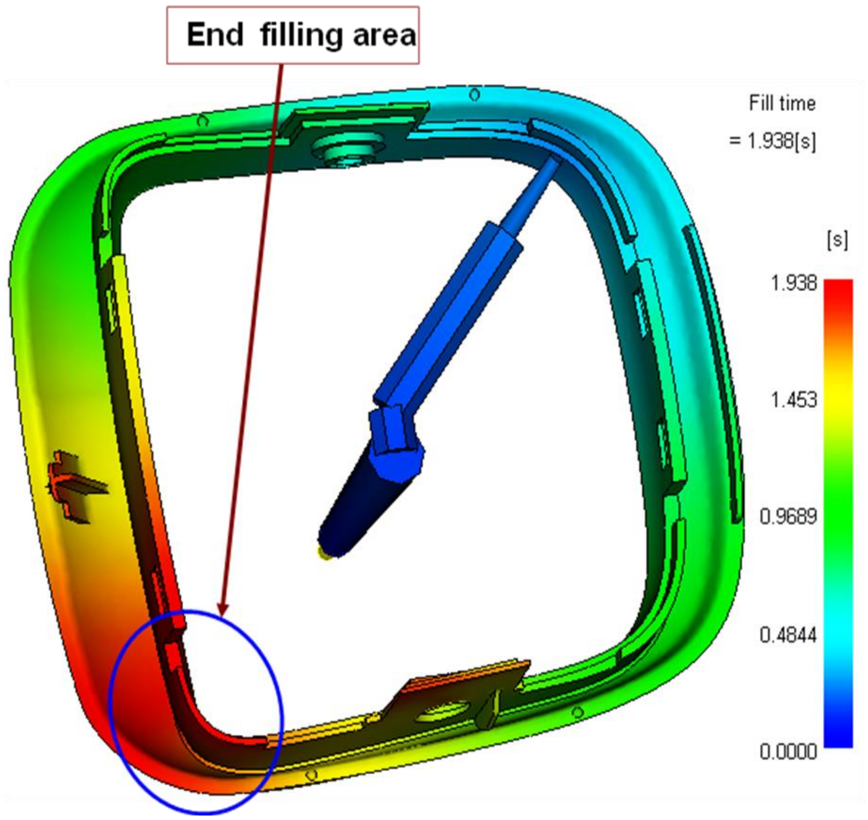

Fig -4: Fill time plot

The maximum shear rate observed during filling is 30523 1/s, which is below recommended limit (<40,000 1/s) for given material grade. The volumetric shrinkage observed in part is in the range of 1-3 % and it is in the allowable range. Maximum injection pressure required to fill the part is 60 MPa. Packing pressure provide 50 MPa for 12 sec.[4]

The sink marks observed on the part is of magnitude 0.0407 mm, these sink mark may be visible on the part

4.5 Weld Lines/ Temperature Overlay

The weld line formed at the end of fill. This is due to merging of two flow fronts. Weld line may be structurally weak as temperature drop is not within allowable limit at one location.[5]

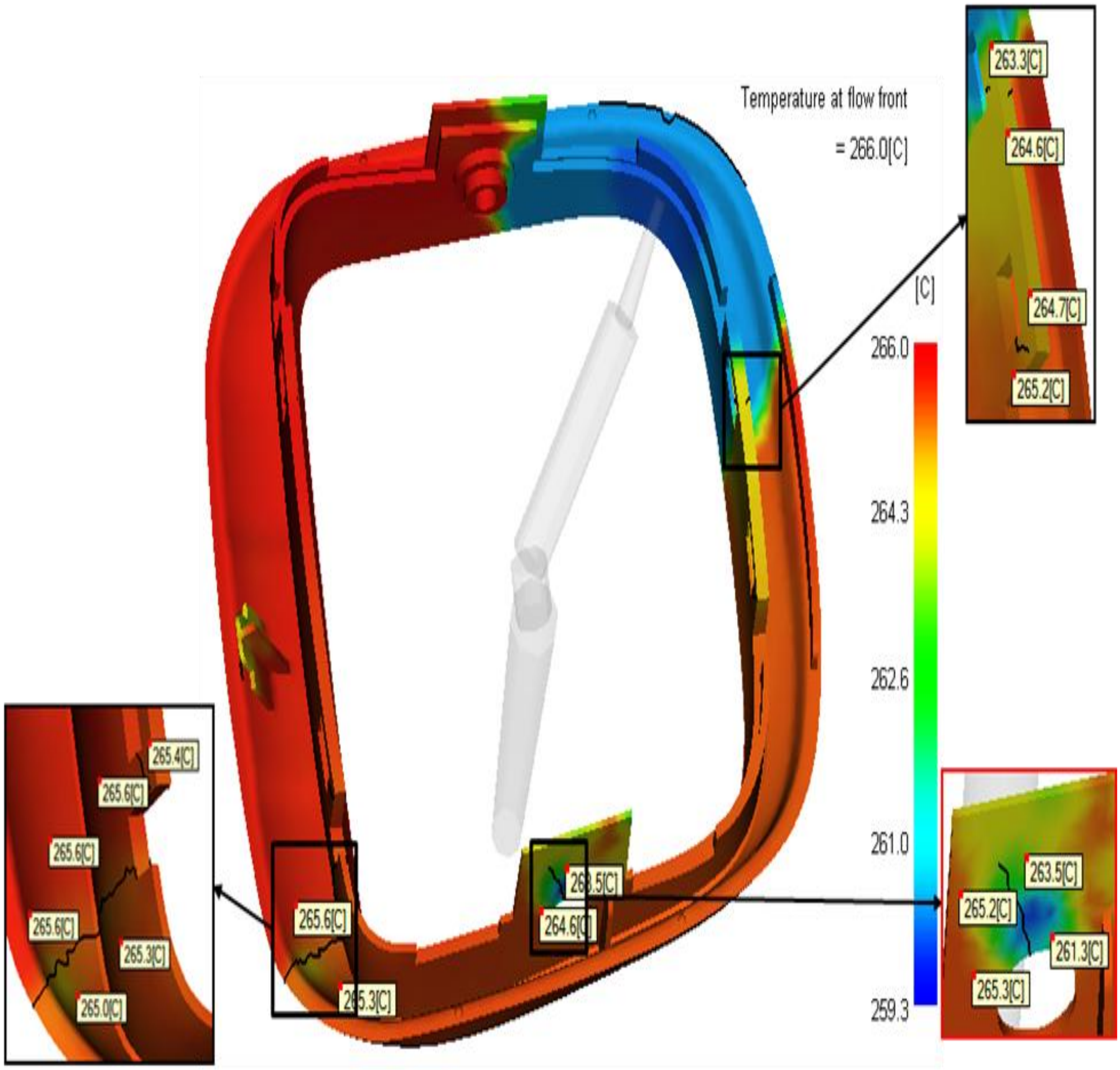

Fig -5: Weld lines plot

5. VALIDATION

The validation of the design will be done by producing the component with the help of the developed mold without affecting the component's functionality. Flow of plastic will be observed. Dimensional accuracy will be measured and checked with the specified dimensions. Visual and actual inspection will be done while attempting to identify the defects. Further, for fitment in the sub-assembly the component will be checked.

6. CONCLUSION

The design and analysis of injection mold for Airvent bezel is successfully fulfills the requirements and various results are obtained after analysis of the part. As the ultimate aim of this study is to find out Weld lines, filling time, cooling circuit etc are nearly achieved.

REFERENCES

[1]. R. G. W. Pye,"Injection Mold Design an introduction and design manual for the thermoplastic industry".

[2]. C.G. Li, Yuguang Wu, "Evolutionary optimization of plastic injection mould cooling system layout design" IEEE 2010 [693-696]

[3]. Zhou Chun-ying, Wang Li-tao, " Injection Mold Design ased on Plastic Advisor Analysis Software in Pro/E," IEEE 2011 [205-208]

[4]. B. Bona, L. Giacomello, C. Greco, A. Malandra,"Position Control of a Plastic Injection Moulding via Feedback Linearization Machine", Arizona December 1992 [2591-2593]

[5]. Chi-Huang Lu and Ching-Chih Tsai,"Multivariable Self-Tuning Temperature Control for Plastic Injection Molding Process," [702-709]

[6]. C. T. Wong, Shamsuddin-Sulaiman, Napsiah Ismail & A.M.S. Hamouda, "Design and Simulation of Plastic Injection Molding Process" Pertanika J. Sci. & Techno. Supplement (2004) [85-99]

[7]. Fauzun, M. Hamdi , A.E. Tontowi, T. Ariga,"Formulation of the size and position of spiral cooling channel in plastic injection mold based on fluent simulation results". IEEE 2008 [1728-1733]

BIOGRAPHIES

Mr. Jitendra Dilip Ganeshkar is currently doing ME in CAD/CAM at Jawaharlal Nehru Engg college Aurangabad, also a lecturer in PCCOE, Pune, Maharashtra, India.
Email:jd_ganeshkar@rediffmail.com
Mob no. +919763013527

Prof. R.B.Patil is the Associate professor in Jawaharlal Nehru Engg college Aurangabad, Maharashtra, India. Their research interest includes CAD/CAM.
Email: rbpatilcadcam@gmail.com