PLASTIC INJECTION MOLD DESIGN FOR AN AUTOMOTIVE COMPONENT 'AIR VENT BEZEL' THROUGH MOLD FLOW ANALYSIS FOR DESIGN ENHANCEMENT

Mr. Jitendra Ganeshkar
ME CAD CAM Pursuing, Department of mechanical engineering,
MGM's Jawaharlal Nehru Engineering College Aurangabad- 431003, Maharashtra, India

Prof. R.B.Patil
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. Mold Design forms the basis of the development work required for producing the desired number of units in a given time frame, with proper mold design a customer can achieve adequate cycle times, part strength and required level of surface finish. Multi-cavity molds can help produce the components in mass quantities in a short duration. The simplicity of the mold is the key to ensure the quality of the component produced and the associated costs of development. A review of the same with ingenious inputs in the design phase would help the Company to achieve its overall objectives. 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 Automotive Industry Component 'Air vent bezel' to study the behaviour 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.

INTRODUCTION

The molding may cause defects and its processing offers a challenge during its development phase. 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.

PROBLEM DEFINITION

A large automotive ancillary industry in Pune has offered my Project sponsoring Company "Advent Tooltech Pvt Ltd" Bhosari, Pune to design the mold & manufacture the intricate component 'Airvent Bezel' with given dimensions & potential material for the component as Polycarbonate and ABS. The Mold needs to be designed keeping the manufacturing constraints in mind. Besides, the Design of the Mold is to be accomplished while keeping the associated costs to the minimum.

The molding is normally associated with molding effects and its processing offers a challenge during its development phase. The cost of the physical mold as well as the processing at the production line using the mold involves high cost of rectification and supervision. The criticality of the component also calls for design challenges for unique features in the mold design to overcome processing defects.

SCOPE OF WORK

- To study the component design with the prospective of a mold designer.
- Generate a layout using 3D modeling tools.
- Design and detailing for manufacturing the mold.

OBJECTIVE OF THE PROJECT

Referring to the scope of the work the following are the objective of the work

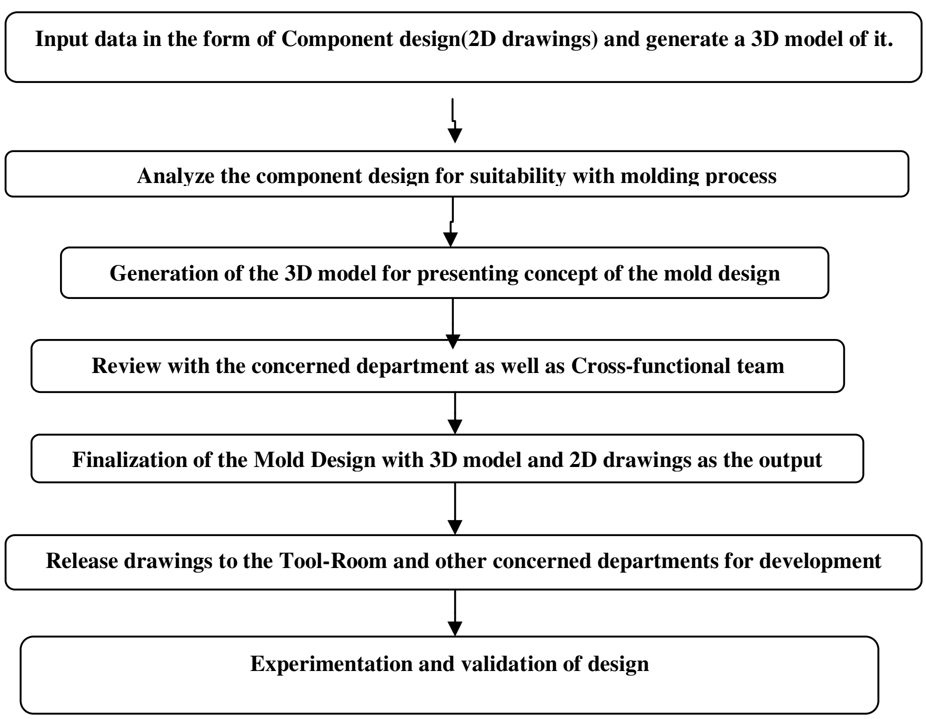
1) Understanding the component design.
2) Identifying the critical locations in the component.
3) Suggest the modifications in the component without affecting its functionality to simplify the mold with smooth flow of molten plastic.
4) Modification in the mold design accordingly.
5) Manufacturing the mold as per new design.
6) Validation of mold by producing the plastic component.

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.

PROPOSED FLOW OF WORK AND METHODOLOGY



EXPERIMENTATION

The draft analysis and/or the flow analysis of the component would provide useful inputs for anticipating the performance of the component during its processing phase. It is generally not feasible to generate a soft mold for experimentation because of high cost involved.

Experimentation of mold design will be done by varying the parameters like type of gate, gating system location, venting location and location of runners and risers for producing the defect free component. These parameters will be changed at least in three levels and appropriate experimentation method will be followed.

From the simulation and analysis, Moldflow software provides sufficient information regarding its filling time, injection pressure and pressure drop. With these results, users can avoid the defect of the plastic in actual injection such as sink mark, hesitation, air traps, and overpacking. The analysis will also help the mould designer to design a perfect mold with minimum modifications and which will also reduce the mold setup time. With this analysis and simulation, it will help to reduce time and cost.

DEFECTS IN EXISTING SYSTEM

While producing the plastic part by Injection molding directly, it may have some defect as follows:

- Dimensions not exact
- Sink mark on part
- Air trap
- Weld lines are visible
- Shrinkage of part etc

Also sometimes it happens that due to incorrect clamping force, the mold get damaged. Cooling channels are not properly remove the heat.

In this project work, these defects are tried to eliminate/minimize with the help of 'Mold flow analysis software' to design the mold for the plastic part 'Air Vent Bezel'.

MODELING & ANALYSIS SOFTWARE USED

CATIA V5 R19

CATIA having various modules such as Sketcher module, part design module, assembly module, wire frame and surface design module, drafting module, sheet metal design module. For component Air vent Bezel Surfacing with sketcher module and drafting module used.

SURFACING MODULE

The Surfacing module is used to create part with less thickness and with given dimensions. Initially sketch of component is drawn with proper dimensions by using sketcher module and then converted it into solid by Surfacing module with various toolbar such as Bound, Extrude etc.

DRAFTING MODULE

To do the documentation of the part created in the form of engineering drawing views and their detailing, the Drafting module is used.

There are two types of drafting:

1. Generative drafting
2. Interactive drafting

The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies, the parametric dimensions added to the component in the part of design workbench during its creation can also be generated and display automatically in the drawing views. The generative drafting is bi-directionally associative in nature. User can also generate the bill of material and balloons in the drawing views in interactive drafting, user needs to create the drawing views by sketching them using the normal sketching tools and then adding the dimensions.

MOLD FLOW ANALYSIS

Mold Flow software provides tools that help manufacturers predict, optimize, and validate the design of injection molds. Companies worldwide use Autodesk Simulation Moldflow Adviser software to help reduce the need for costly physical prototypes, reduce potential manufacturing defects, and get innovative products to market faster, providing a wide range of injection molding simulation tools to help CAE analysts, designers, engineers, mold makers, and molding professionals create more accurate digital prototypes and bring better products to market at less cost.

In Mold Flow software initially want to import a CAD model, then provide proper meshing to that CAD model. After that inspection of the model grid defects carried out. Then set molding process, material selection, setting gate location, analyse model and then view result of analysis.

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.

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 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.

THE DIMENSIONS OF AIR VENT BEZEL WANT TO MANUFACTURE:

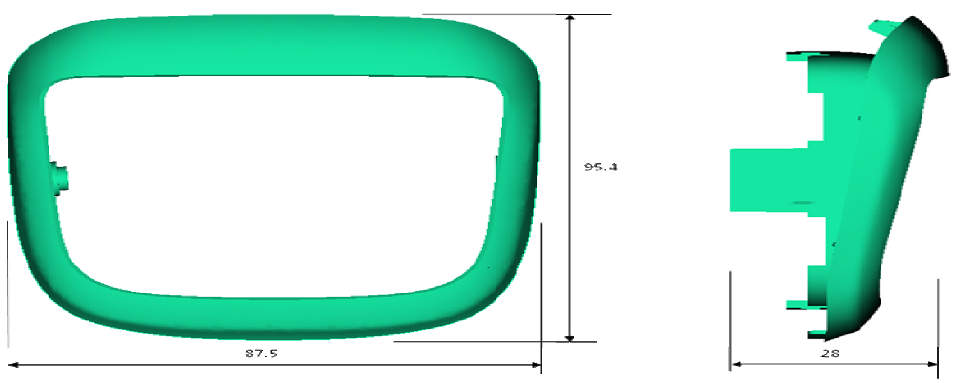


Fig : CAD Model of Air vent Bezel

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

$$\text{Hence, Projected area} = 3.44 \times 3.7$$
$$= 12.9 \text{ inch}^2$$

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.

$$\text{Clamp force required} = \text{Projected area} \times \text{Clamp factor}$$
$$= 12.9 \times 5$$
$$= 64.5 \text{ Tons}$$

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:

$$\text{Clamp force required} = 64.5+6.45$$
$$= 70.95 \text{ Tons}$$

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

$$\text{Clamp force required} = 70.95+7.095$$

ISSN No - 2394-3696

= 78.045 Tons

The nearest machine size to that requirement is 100 Tons.

MOLD FLOW 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, Bay blend T65 XF, Bayer material science.

MESHED MODEL

For Air Vent Bezel Tetra meshing is used as Part is complex and wall thickness is no uniform. Also node to node connectivity is good in Tetra meshing and it is less time consuming and give accurate results. Tetra meshing is available in Mold Flow analysis software

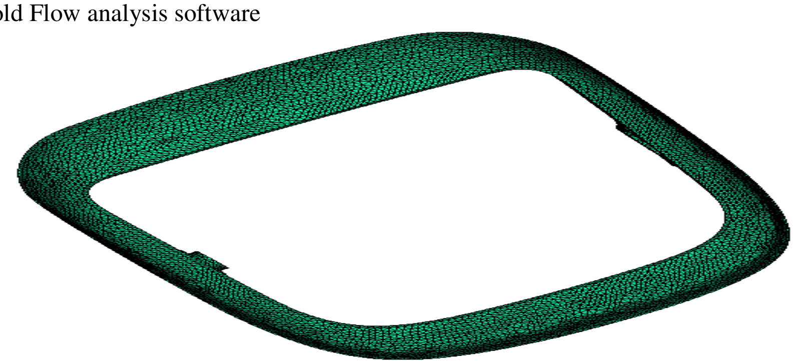


Fig: Meshed model

```
Entity counts:
    Tetras:          297419
    Connected Nodes:         59599

Volume:
    Tetra:      16.2483 cm^3
    Total:      20.2948 cm^3

Aspect ratio:
                  Max     Average    Min
                  26.0      3.24     1.09
```

```
Total number of elements                         =     297431
  Number of part elements                        =     297431
  Number of tetrahedral elements                 =     297419
  Number of sprue/runner/gate elements           =         12
```

PROCESS PARAMETERS USED

Following parameters are used for analysis

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

COOL ANALYSIS RESULTS

The circuit for cooling the mold is important to maintain temperature inside the mold. The Circuit Flow Rate is 1.525 Lit/min. The analysis results of cooling are as follows

Table: 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)

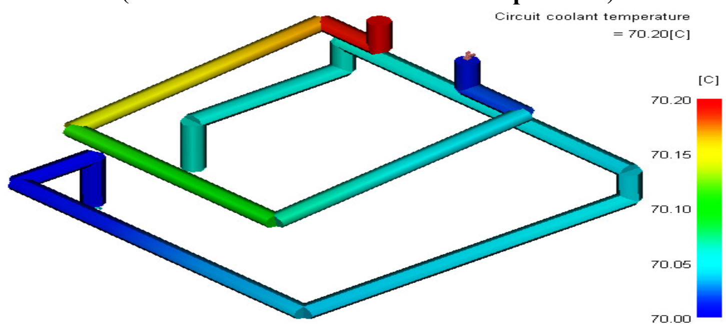


Fig: Circuit coolant temperature

CIRCUIT HEAT REMOVAL EFFICIENCY:

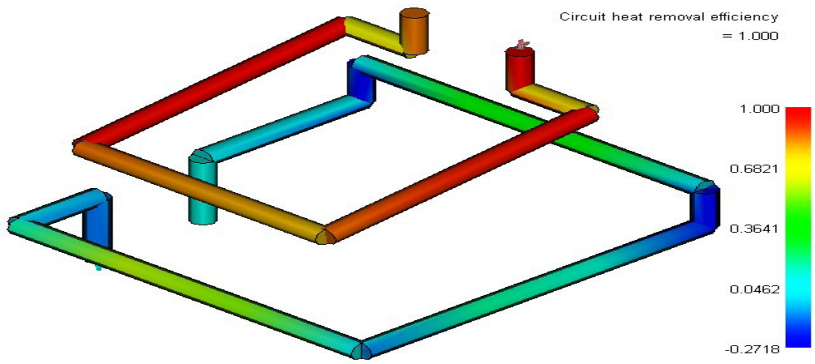


Fig: Circuit heat removal efficiency

Red color indicate 100 % heat removal efficiency from mould. Blue color shows 27 % heat removal from the mould.

CIRCUIT PRESSURE:

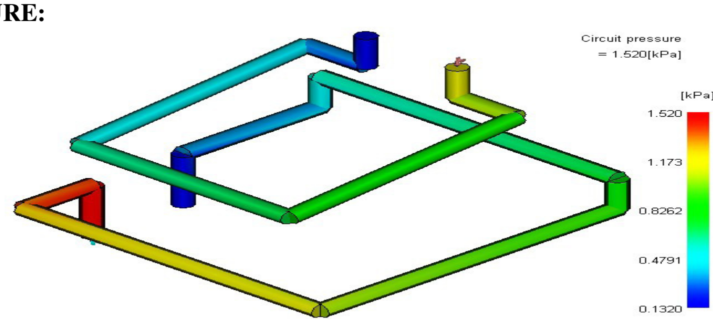


Fig: Circuit Pressure

Circuit pressure observed in cooling channel is 0.132 kPa to 1.52 kPa.

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

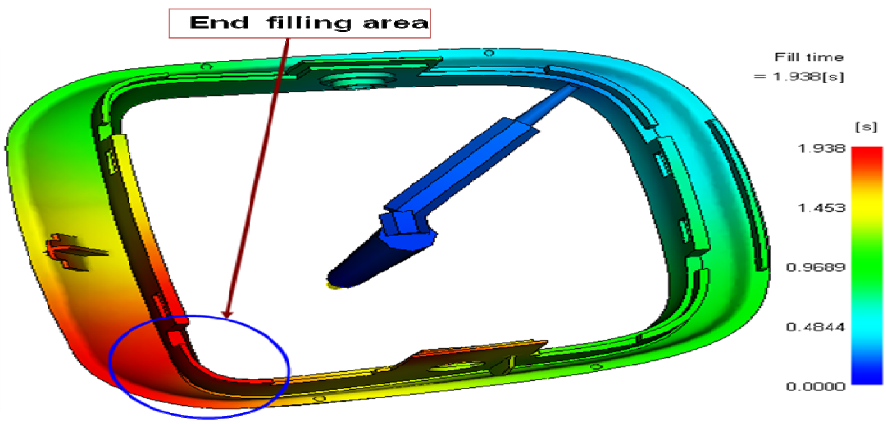


Fig : Fill time plot

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.

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

TIME TO REACH EJECTION TEMPERATURE

Average time to reach ejection temperature for the part 25 sec. packing pressure provided for 12 sec. gate freezing time 12 sec.

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.

AIR TRAP PLOT

Fig: Air trap

Blue color dots shows air trap location. Venting should be provided above locations

RESULT COMPARISON

The actual dimensions of the Air Vent Bezel which are required are compared with the part produced after mold flow analysis data, these dimensions are in the acceptable range.

Following table shows the result comparison between required dimensions & dimensions of produced part for 3 number of sample:

ISSN No - 2394-3696

Table: Result comparison

Parameter	Specified size	Test sample 1 Reading	Test sample 2 Reading	Test sample 3 Reading
Thickness	2 (+0.12/-0.1)	2.107	2.062	2.149
Length	87.5(+/-0.4)	87.156	87.267	87.882
Breadth	28(+/-0.2)	28.068	28.172	28.123
Height	95.4(+/-0.2)	95.402	95.38	95.596

Also the is no any defects in produced part such as sink mark, air trap, weld line and produced part are acceptable.

Image: Actual photograph during working

VALIDATION

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

REFERENCES

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

[2] 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]

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

[4] Darshan P. & Ramesh Babu. K, "Mold flow analysis for PH meter cover", IJMPE, 2013

[5] J. Aisa, C. Javierre, J.A. De la Serna, "An example of simulation tools use for large injection moulds design: The CONTENURTM 2400 l solid waste container", Elsevier JOMPT 2006, [15-19]

[6] M. Dastagiri , M. Prasad and M Annamacahrya , "Application of Axiomatic Design in injection molding process," IEEE 2010 [222-228]

[7] Mold flow design GUIDE BOOK by Shoemaker

[8] Processing data for the injection molder by Bayer MaterialScience Polycarbonates Business Unit, D-51368 Leverkusen, Germany

[9] R. Dubay, B. Pramujati and J. Hernandez, "Cavity temperature control in plastic injection molding, IEEE July 2005 [911-916]

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

[11] Simulation Moldflow Information by Autodesk

[12] Yung Kang Shen, Chih Wei Wu, Ya-Feng Yu, Hung-Wei Chung, "Analysis for optimal gate design of thin-walled injection molding", Elsevier ICIHAMT 2008, [728-734]

[13] Z. Rutkauskas, A. Bargelis, "Knowledge based method for gate and cold runner definition in injection mold design", MECHANIKA 2007