FAMILY MOULD DESIGN FOR QUALITY LEVERS

Ebinezar[1], Bheemrao Kamble[2]

[1]*Asst.professor, BKIT Bhalki, Karnataka, India*
[2]*Asst.professor, BKIT Bhalki, Karnataka, India*

Abstract

Moulding is a process in which by pouring a molten metal into a mould cavity which is created by keeping the required shape of pattern on the molding sand and allowing the molten metal to solidify by cooling slowly. But in this process we can able to get only one shape of casting, it means shape of the final casting is limited. If we want more number of same design but not identical parts then it is not possible from a single mould, this drawback can be overcome by family moulding. This paper tells how can be able to get more number of same design but not identical parts. In another words we can say that two different parts can be produced from a single mould. This mould is called family mould. Family mould design consists of a complete mould with a few or many different cavity inserts that runs as a multiple part. Hence Family mould design for plastics levers is made here by using injection moulding machine for injection of molten plastic into mould cavity and UGNX5 software for injection mould design for plastic levers.

Keywords: *Injection Moulding, Pattern, Solidification, Casting, Molten*

---***---

I. INTRODUCTION

In the injection moulding process the injected polymer flows through the sprue, runners and gate to the cavity (or cavities). The polymer flow in mould channels is unsteady and non isothermal and depends on many factors, like the properties of polymer used, injection mould design (especially cavity shape and runners configuration), injection moulding machine and processing conditions. [1]The polymer and the machine are usually the factors determined by moulded parts' manufacturers and consumers. The melt flow in mould runners can be controlled by the design and manufacturing technology of the mould as well as by the processing conditions in order to obtain the moulded parts with expected morphology, properties, shape, dimensions and surface. [2]The cavity in the injection mould should be filled totally during the injection phase and the way of filling should be The selection of most appropriate design and technological solutions to produce certain mould should capture technical performance, economical issues as well as environmental impacts occurred during the mould life cycle. The selection of alternative mould design solutions in the early design stage It includes the use of Life Cycle Assessment methodology, Life Cycle Cost methodology and is supported by numerical simulations. [3]Assuring the proper mould temperature for a specified polymer is a very important issue as well as keeping the temperature constant and equal across the cavity surface. Differences in mould temperature can lead to the problems with manufactured parts like warpage. [4]Injection molding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost. Its complexity and the enormous amount of process parameter manipulation during real time production create a very intense effort to maintain the process under control. What is more, complexity and parameter manipulation may cause serious quality problems and high manufacturing costs [5]. One of the main goals in injection molding is the improvement of quality of molded parts besides the reduction of cycle time, and lower production cost.

For design an [6-7] injection mould for plastic levers of HVAC the mould has to be designed to the ease of manufacturability, assembly and positive ejection of the component within the minimum possible time.

HVAC is an abbreviation for heating, ventilation, and air conditioning system-the technology of indoor or automotive environment. In a vehicle heat from the engine raises temperature inside the vehicle. [8-10]Ventilating is the process of replacing air in vehicle space to control temperature or remove moisture, carbon dioxide and to replenish oxygen. Ventilation includes both the exchange of air to the outside as well as circulation of air with in the vehicle. Air conditioning means removal of heat from the vehicle using a refrigerant. Plastics are the high molecular weight polymers. Now a days to get formability, consolidation of the parts and providing a low cost to performance ratio it is necessary to make plastic castings.

1.1 Hvac

It is abbreviation for heating, ventilation, and air conditioning system-the technology of indoor or automotive environment. Heating In a vehicle stands for raising the temperature inside the vehicle using heat heating from the engine. Ventilating It is the process of process of replacing air in vehicle space to control temperature or remove moisture, odors, smoke, heat, dust, airborne bacteria, carbon dioxide and to replenish oxygen. Ventilation includes both the exchange of air to the outside as well as circulation of air with in the vehicle. Air conditioning: It means removal of heat from the vehicle using a refrigerant.

1.2 Family Mold Design

Family mold is usually built to save money on multiple part types. Family mold consists of a complete mold with a few or many different cavity inserts that runs as a multiple part each cycle Family molds are often utilized when two or more part designs are similar but not identical, or if mold cost is a driving factor.

Two different parts can be produced from a single mold. This can save on time and expenses by sharing common mold components such as the mold base, However, family molds are typically better suited for lower volume applications, and automation may be necessary to separate pieces during or after production. They also have the potential for greater downtime as repairs and modifications to a single part affect all the components within the mold. Some of the benefits of a family mold are lost if the parts in the mold are run in different resins. Still, with today's cost conscious buyers desire for versatility family molds is becoming an increasingly popular design option.

A family mold is nothing but a multi-cavity injection mold in which each cavity forms a component part of the finished product. You can create an individual core and cavity, and individual inserts, for each plastic part. Below fig1 shows a typical family mold.

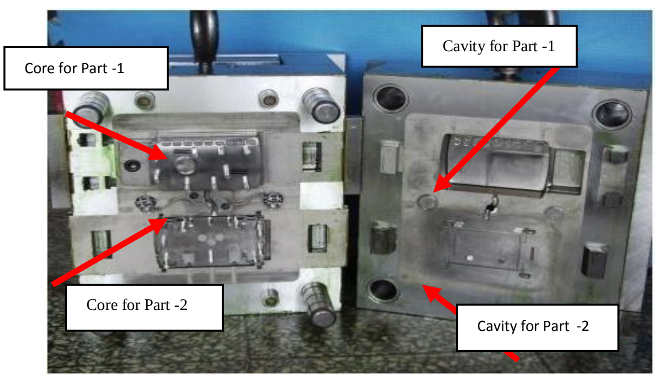

Fig1. Typical family mold

1.3 Shrinkage and Warpage

All materials when heated expand and on cooling contract. The method of injection moulding also is subjected to expansion of plastic during injection phase and its contraction during cooling for solidification, Hence the volume change from mould dimension to the Part dimension causing shrinkage of plastic. shrinkage of plastic depends on material and its processing.

Various factors that affect shrinkage are

1. Basic plastic material
2. Filler used and percentage
3. Part wall thickness
4. Melt temperature
5. Mould temperature
6. Injection time

2. METHODOLOGY OF MOLD DESIGN

2.1 Methodology of Mold Design

- Check the model feasibility to design.
- Identify critical dimensions and method to achieve these dimensions.
- Prepare dimension sheet.
- Prepare 3D model of component based on tool MMC.
- Decide Parting Line
- Add drafts to the model considering tolerances for maximum material condition.
- Add shrinkage to the model.
- Decide gate location for complete fill and good part quality after analysis.
- Design feed system.
- Plan for efficient cooling/heating circuits.
- Plan for ejector positions
- Plan core and cavity inserts.
- Plan for side core.
- Internal splitting of core and cavity inserts considering manufacturing and assembly requirement.
- Use standard elements wherever possible.
- Decide proper size mold base.
- Prepare the concept design.
- Conduct the concept design review meeting.
- Change the mould design based on the review output.
- Freeze the mould design.
- Prepare manufacturing drawing and bill of material
- Prepare detail drawing of inserts and mold base elements.
- Release 2D and 3D drawings to manufacturing.

2.2 Component Details

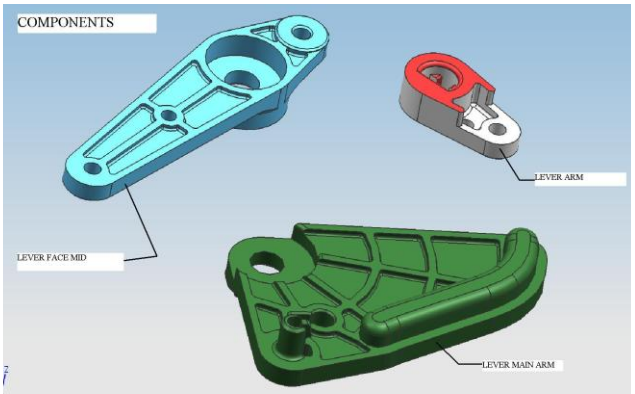

Fig- 2: Levers

Component Name: Levers
Components Weight: 30 gms.
Material: POM 20% TF
Mold Shrinkage (mm/mm):0.02
Nominal Wall thickness: 2 mm
Type of Gate: Edge Gate
No. of Cavities: 1
Type of Tool: Family mold
Production requirement: 2200 components/day.

2.3 Mold Feasibility Study

1. Minimum wall thickness is more than 0.6 mm -----Yes
2. Free from Sudden drop of wall thickness at rear to gate area because it is restriction to fill----- No
3. Ribs thickness must be lesser than joining wall, ideally this should be 60% of basic wall thickness-----No (It is maintained of 80% of basic wall thickness)
4. Ejection possibility (auto/semi auto) ----- Yes
5. 3D model matching w.r.t 2-D drawing ------ Yes
6. Is part catching in core side (ejection half)----- Yes

2.4 POM Material Specifications

Specification	
Density (gm/cc)	1.4
Mold Shrinkage (mm/mm)	0.02
Melt temperature	180 – 230 °C
Injection Speed	Medium
Injection Pressure	700-1400 kg/cm^2
Pre-Drying Conditions	3 hrs at 90 – 100 °C
Total Heat Content	110 cal/gm
Thermal Conductivity of Plastic	$5.5x\ 10^{-4}$ cal/sec.cm
Heat distortion Temp.	140 °C
Mold temperature	60 – 100 °C

2.5 Gate Location and Fill Time

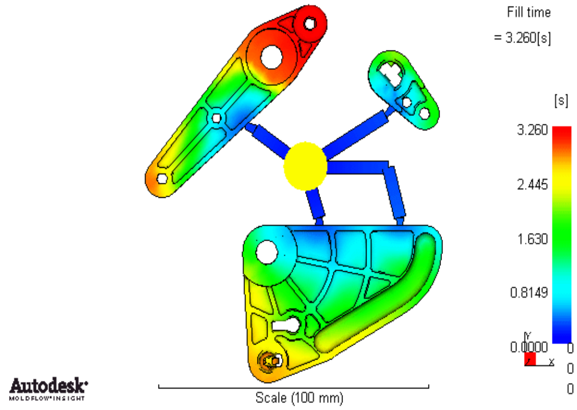

Fig. 3 Fill analysis

The Injection point or the gate location is centre of components as shown in fig. The feed system for this component is cold runner. The fill analysis shows a different injection point than the actual selected injection point. This is because to accommodate the selected cold sprue into the design based in the cavity. The filling pattern for the component in the new selected point is also similar to that of the analyzed point because there is no much major change in the injection point.

2.6 Weld Lines

The weld line formed in the coloured area and is inevitable. The weld line would be strong as the fusion temperature is well within the material specified temperature limits.

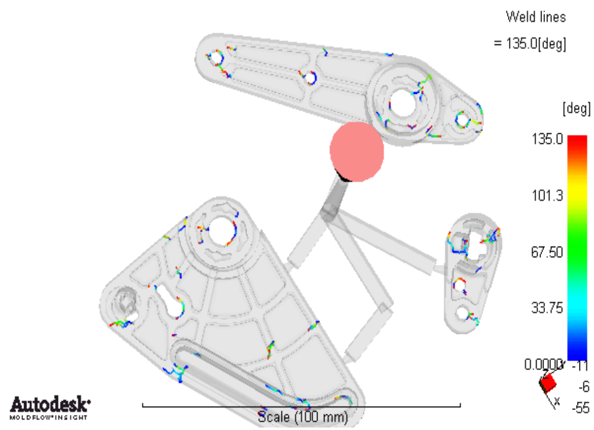

Fig- 4:.weld lines

2.7 Sink Marks

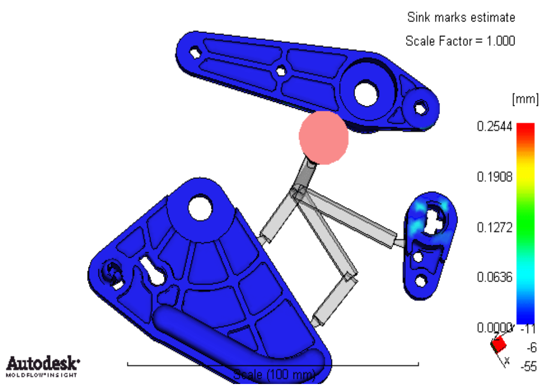

Fig- 5: Sink Marks

2.8 Volumetric Shrinkage

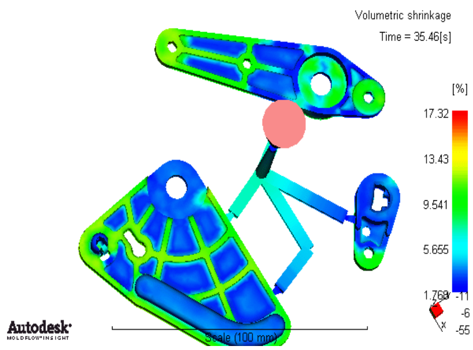

Fig-6: Volumetric Shrinkage

3. DESIGN CALCULATIONS

3.1 Mould Machine Selection

Selection of machine is mainly based on three factors.

- Based on clamping tonnage required
- Based on shot capacity
- Based on plasticizing capacity

Weight of components is 30 gms from cad model.

Based on clamping tonnage required

Clamping force Fc = Pc x {(A_p x no. of cavities) + AR}

Where,

Pc = Cavity pressure in Kg/sq.cm.
Ap = Projected area in sq.cm.
AR = Runner area in sq.cm.
PI = Injection pressure in Kg/sq.cm.
Pc = (1/2) PI for easy flow materials or (1/3) PI for viscous materials.

Therefore, Pc = 1/3 x1000
= 333.33 kg/cm^2
A_p = 62.0 cm^2
A_r = 14.30 cm^2

Clamping Force = 333.33 x 76.30
= 25433.33 kgf
= 25.43 Tonnes

Based on all the above requirements J85AD-110H

3.2 Machine Details

- Shot Capacity for POM= 105 gm
- Injection rate = 337 cm^3/sec Plasticizing rate for PS = 92 kg/hr
- Screw Diameter = 35 mm
- Clamping Force = 110 Tonnes
- Distance between Tie bars = 410x360
- Mould Opening Stroke = 300 mm
- Mold Height = Min180, Max 510
- Day Light = 810 mm
- Maximum Ejection Stroke = 80 mm
- Machine Platen Size = 580x 530

3.3 Estimation of Cycle Time

Cycle time = Fill time + solidifying time + Mold opening and closing time + Ejection time

1. **Fill time**

Injection of material into the impression is equal to fill time.

Fill time = 3.26 sec (from mold flow)

2. Solidifying time

Solidifying time is proportional to square of the wall thickness.

$$Ts = \frac{\rho \text{ x a x t2}}{8 \text{ x } \lambda \text{ x (T mat. – T mold)}}$$

Where, ρ = Density of plastic in gm/cca = Total heat of plastic cal/gm
t = wall thickness in cm
λ = Thermal conductivity of plastic cal/cm sec °C
Ts = 10 sec
Solidifying time = 10 sec

3. Mould opening and closing time
Approximately 10 sec.
4. Mould ejection time= 10 Sec (approx.)
5. Packing time=2Sec (from mold flow)

Cycle time = 3 + 10 + 10 + 10+2 = 35sec ------------ Calculated
Cycle time =35.46 sec ------------ Mold flow

3.4 Mold Details

Name of the component	Levers of HVAC
Type of mold	Family mold
No cavities	1
Mold base	296x346
Type of Gate	Edge gate
Material details	POM 20%TF
Cavity	Insert Type
Core	Insert Type
Ejector Type	Pin
Cooling Type	Channel Cooling
Shut Height	240

3.5 Cavity Splitting

The cavity side is made of inserts based on.

- Ease of manufacturing
- Ease of polishing
- Considering the design changes in the component
- Considering the modification of the tool after first shot
- Considering the critical dimensions in the component all the inserts in the cavity side are assembled to the cavity housing and then they are assembled to the cavity housing and then they are assembled to the cavity plate.

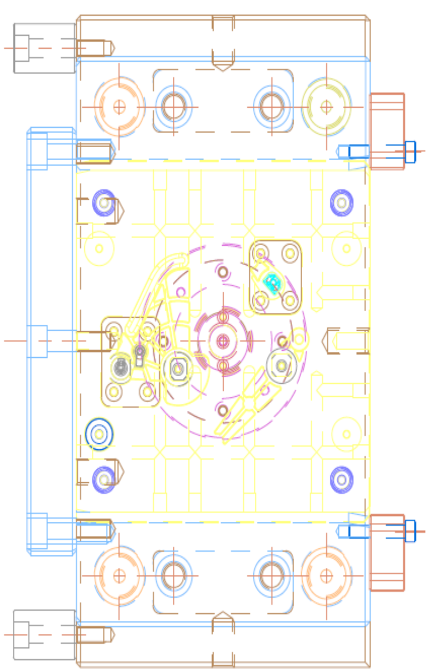

Fig-7: Cavity Splitting Assembly

The Cavity side is split into 6 inserts. The inserts are Main cavity insert, Cavity pin1, Cavity pin 2, Cavity pin3, Cavity sub insert 1, and Cavity sub insert 2. The cavity inserts are split pieces considering the ease of manufacturing. And also shown 2d detailing of these inserts

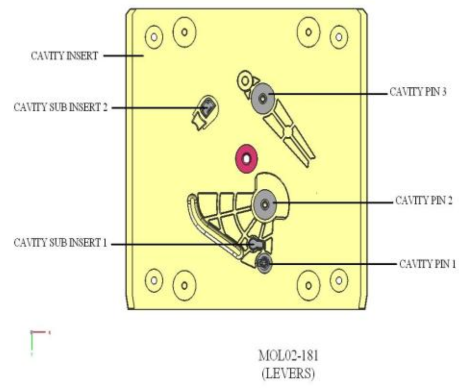

Fig- 8: Cavity insert assembly

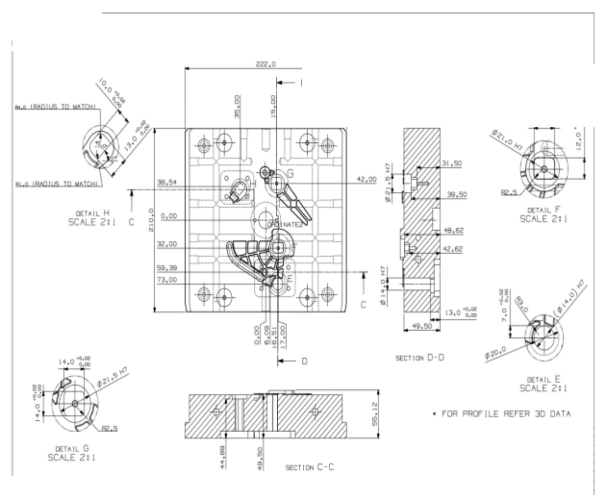

Fig- 9: Main cavity inserts

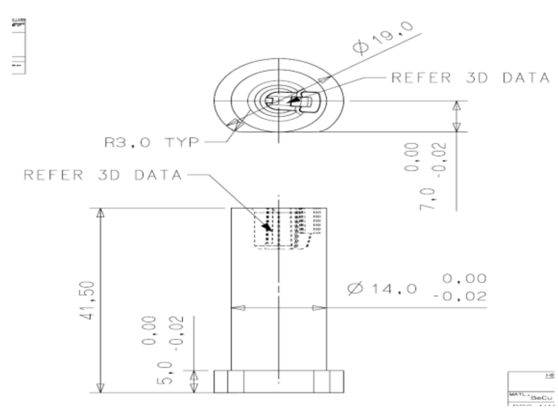

Fig- 10: Cavity Pin-1

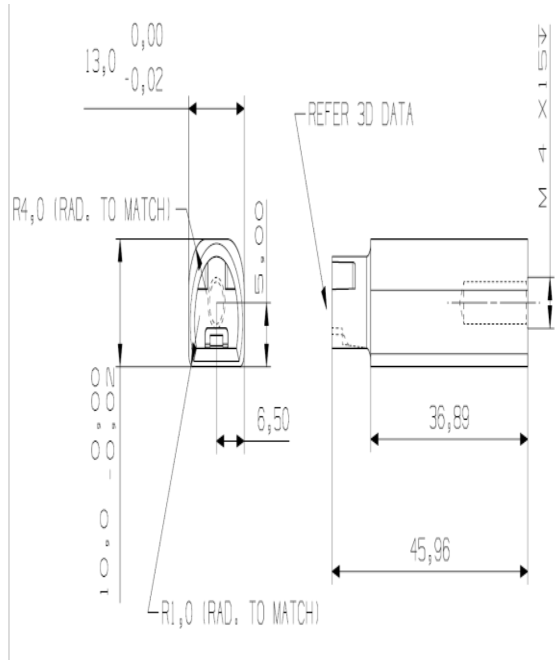

Fig- 11: Cavity sub insert -1

3.6 Core Splitting

The core side is made of inserts based on.

- Ease of manufacturing
- Ease of polishing
- Considering the design changes in the component
- Considering the modification of the tool after first shot
- To avoid the shot filling due to improper venting and ease of providing vents.
- Considering the critical dimensions in the component.
- To avoid the flash due to poor shutoff.
  All the inserts in the core side are assembled to the core housing and then they are assembled to the core plate.

4. CONCLUSIONS

Family moulding is one of the most common methods used to process plastics. We have to consider various factors to produce a defect free and economical plastic component during the design stage itself. Flexibility of shape of the final casting in a single mould is achieved. Less effort of workers and time saving can be done hence in shorter time with less efforts of workers high efficiency involved quality casting can be produced since family moulding process is very fast. Hence production rate is more. When production rate is more then it shows minimised defect castings are manufacturing hence economical.

REFERENCES

[1]. Gasper Gantar, Andrej Glojek , Mitja Mori, Blaz Nardin, Mihael Sekavcnik " journal of Mechanical Engineering" 59(2013)3, 193-200.
[2]. E. Boci¹ga, T. Jaruga,"Experimental investigation of polymer flow in injection mould", International Scientific Journal Volume 28, Issue 3,March 2007,Pages 165-172.
[3]. E. Bociga, T. Jaruga, K. Lubczynska, A. Gnatowski " Warpage of injection moulded parts as the result of mould temperature difference" International Scientific Journal Volume 44, Issue 1,March 2010,Pages 28-34.
[4]. P.K. Bharti M. I. Khan " Recent methods for optimization of plastic injection molding process- a retrospective and literature review" International Journal of Engineering Science and Technology Vol. 2(9), 2010, 4540-4554.
[5]. Nagsen B. Nagrale , Dr.R.N.Baxi "Finite Element Analysis of Reciprocating Screw for Injection Molding Machine" International Journal of Engineering and Technology Vol.3 (3), 2011, 191-199.
[6]. Alban Agazzi , Vincent Sobotka , Ronan LeGoff , Yvon Jarny. "Optimal cooling design in injection moulding process -A new approach based on morphological surfaces" applied thermal engineering 52 (2013) 170-178.
[7]. OYETUNJI, A. "Development Of Small Injection Moulding Machine For Forming Small Plastic Articles For Small-Scale Industries" Journal of Engineering Science and Technology Vol. 5, No. 1 (2010) 17 – 29.
[8]. Donald V. Rosato, "Injection molding handbook"
[9]. Dlenviel and Denton, " Fundumentals of injection Moulding".
[10]. Oswal, "Injection molding hand book".