Review on Tool Design and Analysis of Injection Moulding

M.B.Pagar[1], P.G.Scholar, Department of Mechanical Engineering, Amrutvahini College of Engineering , Sangamer, pagar.mahesh@gmail.com
V. Wakchaure[2], Assistant Professor, Department of Mechanical Engineering, Amrutvahini College of Engineering , Sangamer, wvishnu@gmail.com

Abstarct-Injection molding is a manufacturing process for producing plastic parts from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mold cavity by a reciprocating screw or a ram injector, where it cools and hardens to the configuration of the mold cavity The mold is usually constructed using either steel or aluminum, and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including computer components to outdoor furniture.

In injection parts manufacturing, tooling is very important. The quality of the product of compound depends on the tool by which it is manufactured. In this case injection molding tooling is used. If tool design properly it effects the quality of the compound .It can lead to shrink marks, pore finish, short shots , wrapping ,bubbles formation in parts .Such problem arises when injection mold is not design properly .

This seminar address the method of design and analysis of injection molding, analyzing the thermal stresses, injection pressure and other parameter are studied.

1. INTRODUCTION

Injection molded components are consistently designed to minimize the design and manufacturing information content of the enterprise system. The resulting designs, however, are extremely complex and frequently exhibit coupling between multiple qualities attributes. Axiomatic design principles were applied to the injection molding process to add control parameters that enable the spatial and dynamic decoupling of multiple quality attributes in the molded part. There are three major benefits of the process redesign effort. First, closed loop pressure control has enabled tight coupling between the mass and momentum equations. This tight coupling allows the direct input and controllability of the melt pressure. Second, the use of multiple melt actuators provides for the decoupling of melt pressures between different locations in the mold cavity.
Such decoupling can then be used to maintain functional independence of multiple qualities attributes. Third, the heat equation has been decoupled from the mass and momentum equations.

2. LITERATURE REVIEW

REVIEW FROM PAPERS

2.1. The experimental study on the defects occurrence of SL mold in injection molding

Chao-Chyun et al. (2008)

In this paper they have shown the different defects occurrence in injection molded parts of metal including weld line, flow mark and solid skin. In this study , a thin wall cavity is designed as flow path for plastic injection mold. The injection molding tests were performed by using metal mold and stereolithography mold to compare with the flow behavior and defects occurrence of flat parts. The experiment was perfumed with various process parameters to investigate the defects occurrence in Injection-molded parts.

2.2 Research on failure and material selection of Plastic mold

Bohai He.(2008)

This paper is about material selection and heat treatment of the plastic mold. The main failure modes of the plastic mold surface wear, deformation and fracture. The main failure reasons depend on working condition, mold materials and heat treatment. The working conditions of the plastic mold are complex. They touch directly with the plastic , withstand the pressure , temperature , frication and corrosion and so on.

ISSN : 2394-0905

2.3 **Investigation on the welding strength of thin-wall injection molded ABS parts**

Chun-Sheng et.al (2007)

This paper investigates the influence of processing conditions on the weld line strength of thin-wall Acrylonitrile Butadiene Styrene Copolymer (ABS) parts. It is well know that the wildling reduces the mechanical performance of the conventional injection molded parts. Yet systematic researches and reports on wildling strength of thin-wall molded parts are still insufficient.

2.4 **Computer simulation of transport processes during injection mold-filling and optimization of the modeling conditions**

Amit Kumar et.al (2002)

This paper give details with the computer simulation of injection mold-filling at flow rate during the production of a cylindrical part under isothermal and non-isothermal conditions. The material of the object is low density polyethylene (LDPE) power-law viscosity of LDPE is also a function of temperature. The material of the mold is steel. The concept of melt-mold thermal contact resistance coefficient has been incorporated in the model for the non-isothermal filling conditions.

2.5 **Plastic mold design of top-cover of out-shell of mouse based on CAE**

Jain ZHOU et.al (2011)

In this paper analysis the product construction and manufacturing technology ,aso points parting surface design, the cavity mold design and the punch mold design , calculating deformation parts working size ,drawing of assembly the parts graphs, manufacturing process of plastic mold. The product is the top-cover of out-shell of mouse. Firstly ,it should have some comprehensive mechanical properties , including good mechanical strength ,stable one. From the economic , using and forming property , ABS material can satisfy the all demands.

2. OBJECTIVE OF INJECTION MOLDING

Injection molding is a manufacturing process for producing plastic parts from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mold cavity by a reciprocating screw or a ram injector, where it cools and hardens to the configuration of the mold cavity. After a part is designed, usually by an industrial designer or an engineer, molds are then manufactured by an injection mold company, where it is assigned to a mold maker (or toolmaker). The mold is usually constructed using either steel or aluminum, and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including computer components to outdoor furniture.

3. INJECTION MOLDING PROCESS

Injection molding is a manufacturing process for producing plastic parts from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mold cavity by a reciprocating screw or a ram

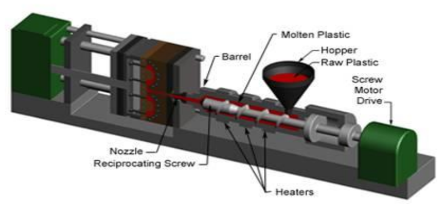

Fig.1.1 Injection molding process [11]

injector, where it cools and hardens to the configuration of the mold cavity. The mold is usually constructed using either steel or aluminum, and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including computer components to outdoor furniture.

Injection molding machines have many components and are available in different configurations, including a horizontal configuration and a vertical configuration. However, regardless of their design, all injection molding machines utilize a power source, injection unit, mold assembly, and clamping unit to perform the four stages of the process cycle.

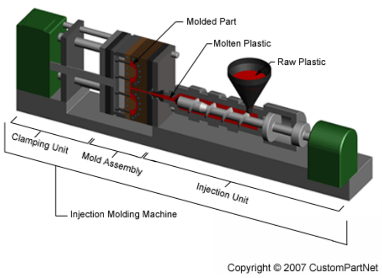

Fig.1.2 Injection molding machine - Injection unit [11]

Injection molding machines consist of a material hopper, an injection ram or screw-type plunger, and a heating unit. They are also known as presses, they hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can exert. This force keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to over 9,000 tons, with the higher figures used in comparatively few manufacturing operations. The total clamp force needed is determined by the projected area of the part being molded. This projected area is multiplied by a clamp force of from 2 to 8 tons for each square inch of the projected areas.

4.1 Injection unit

The injection unit is responsible for both heating and injecting the material into the mold. The first part of this unit is the hopper, a large container into which the raw plastic is poured. The hopper has an open bottom, which allows the material to feed into the barrel. The barrel contains the mechanism for heating and injecting the material into the mold.

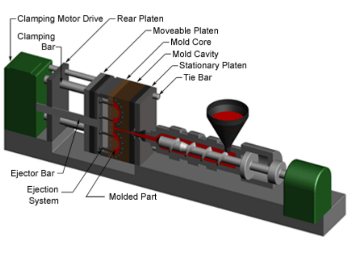

Fig.1.3 Injection molding machine - Clamping unit [11]

4.3 Machine specifications

Injection molding machines are typically characterized by the tonnage of the clamp force they provide. The required clamp force is determined by the projected area of the parts in the mold and the pressure with which the material is injected. Therefore, a larger part will require a larger clamping force. Also, certain materials that require high injection pressures may require higher tonnage machines. The size of the part must also comply with other machine specifications, such as shot capacity, clamp stroke, minimum mold thickness, and platen size.

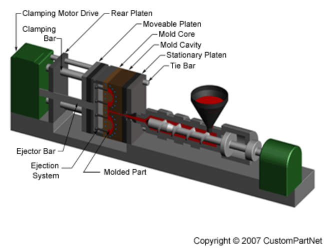

Fig.1.4 Injection molding machine detail view [11]

	Babyplast	Powerline	Maxima
Clamp force (ton)	6.6	330	4400
Shot capacity (oz.)	0.13 - 0.50	8 - 34	413 - 1054
Clamp stroke (in.)	4.33	23.6	133.8
Min. mold thickness (in.)	1.18	7.9	31.5
Platen size (in.)	2.95 x 2.95	40.55 x 40.55	122.0 x 106.3

4.4 Tooling

The injection molding process uses molds, typically made of steel or aluminum, as the custom tooling. The mold has many components, but can be split into two halves. Each half is attached inside the injection molding machine and the rear half is allowed to slide so that the mold can be opened and closed along the mold's parting line. The two main components of the mold are the mold core and the mold cavity. When the mold is closed, the space between the mold core and the mold cavity forms the part cavity, that will be filled with molten plastic to create the desired part. Multiple-cavity molds are sometimes used, in which the two mold halves form s several identical part cavities. [11]

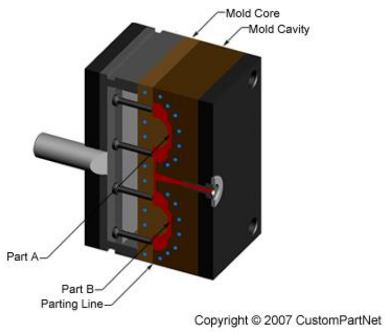

Fig.1.5 Injection molding machine tool process [11]

4.5 Mold base

The mold core and mold cavity are each mounted to the mold base, which is then fixed to the platens inside the injection molding machine. The front half of the mold base includes a support plate, to which the mold cavity is attached, the sprue bushing, into which the material will flow from the nozzle, and a locating ring, in order to align the mold base with the nozzle. The rear half of the mold base includes the ejection system, to which the mold core is attached, and a support plate. When the clamping unit separates the mold halves, the ejector bar actuates the ejection system. The ejector bar pushes the ejector plate forward inside the ejector box, which in turn pushes the ejector pins into the molded part. The ejector pins push the solidified part out of the open mold cavity. [11]

4.6 Mold channels

In order for the molten plastic to flow into the mold cavities, several channels are integrated into the mold design. First, the molten plastic enters the mold through the sprue. Additional channels, called runners, carry the molten plastic from the sprue to all of the cavities that must be filled. These channels allow water to flow through the mold walls, adjacent to the cavity, and cool the molten plastic. [11]

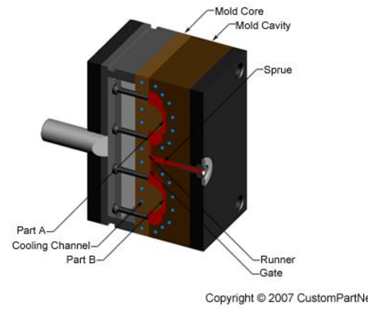

Fig.1.7 Mold channels [11]

.

5. Stages in Injection molding

Stage 1

Granulated or powdered thermoplastic plastic is fed from a hopper into the Injection Molding machine.

Stage 2

The Injection Molding machine consists of a hollow steel barrel, containing a rotating screw (Archemidial Screw). The screw carries the plastic along the barrel to the mould. Heaters surround the barrel melt the plastic as it travels along the barrel.

Stage 3

The screw is forced back as the melted plastic collects at the end of the barrel. Once enough plastic has collected a hydraulic ram pushes the screw forward injecting the plastic through a sprue into a mould cavity. The mould is warmed before injecting and the plastic is injected quickly to prevent it from hardening before the mould is full.

Stage 4

Pressure is maintained for a short time (dwell time) to prevent the material creeping back during setting (hardening). This prevents shrinkage and hollows, therefore giving a better quality product. The Molding is left to cool before removing (ejected) from the mould. The Molding takes on the shape of the mould cavity.

5.1 Types of injection molding process

1. Shell molding
2. Injection molding
3. Compression molding
4. Transfer molding

5. Extrusion molding
6. Blow molding
7. Rotational molding

5.2 Benefits of Injection molding

Injection molding allows for high production output rates.
When producing your product you may use inserts within the mold. You may also use fillers for added strength.
Close tolerances on small intricate parts is possible with Injection Molding.
More than one material may be used at the same time when utilizing co-Injection Molding.
There is typically very little post production work required because the parts usually have a very finished look upon ejection.
All scrap may be reground to be reused, there for there is very little waste.
Full automation is possible with Injection Molding.

5.3 Problems in injection Molding

1. Black specks spots of streaks
2. Bubbles are showing in the finished part
3. Flashing
4. Pore Finish
5. Short Shots
6. Sink Marks
7. Warping
8. Surface Marks

6 .Process characteristics

Utilizes a ram or screw-type plunger to force molten plastic material into a mold cavity
Produces a solid or open-ended shape that has conformed to the contour of the mold
Used to process both thermoplastic and thermosetting polymers, with the former being considerably more prolific in terms of annual material volumes processed. The prevalence of thermoplastics is a result of many factors including:
simultaneous economic, engineering, and manufacturing feasibility
unique and diverse material properties, well suited for a larger proportion of consumer and engineering applications
the ability to soften and flow upon heating facilitates recycling post consumer waste, reject parts, and the melt delivery system
reduces manufacturing risks associated with the injection unit. Thermosets, which cannot flow after the chemical crosslinking occurs, are susceptible to crosslinking in the injection barrel and encapsulating the screw and check valves leading to flow obstructions, mechanical seizures, damage to expensive components, and timely/costly down-times during the mechanical removal of thermosetting polymer from the injection components.
reduces manufacturing time per part produced in many scenarios because the thermal solidification of thermoplastics be achieved in a shorter time duration than the chemical solidification of thermosets.

7. CASE STUDY

The analysis the product construction and manufacturing technology,aso points out the parting surface design,the cavity mold design and the punch mold design, calculating deformation parts working size, drawing of assembly and parts' graphes, manufacturing process of plastic mold .

The product is the top-cover of out-shell of mouse.Firstly,it should have some comprehensive mechanical properties,including good mechanical strength,stable electrical property,and chemical performance.From the material's origin and cost,ABS is the suitable one.From the economic,using and forming property,ABS material can satisfy the all demands.

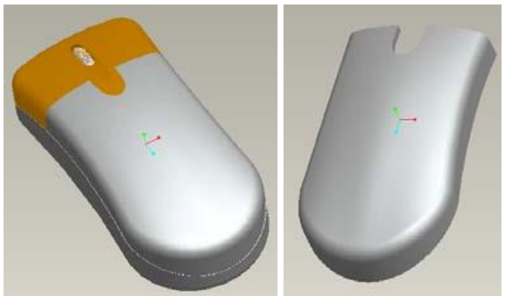

Fig . Upper cover of mouse

According to the principle of parting surface choice,from the comprehensive analysis,the two plans are selected:one parting surface style;two parting surfaces style.

While the two parting surfaces style has such characteristics:the material inputting smoothly and evenly.but the mold construction is complex,the mold thickness is increased,it spends more.The one parting surface can make the mold construction simple,decrease the mold plate thickness,and let the raw material little,the distance of inputting material strip also has been shortened.

From the contrast of two plans,the style of one parting surface have more advantages than the style of two parting surface,and it also can reduce the mold’s cost,so the mold adopts the style of one parting surface. [5]

8. CONCLUSIONS

Mold design is very important part of tooling .Proper tool design is necessary to get perfect and good product, which is required in Automobile industry. In this seminar we have studied type of mold design, problems in mold design , solution for mold design of the product. In mold design or tooling different parameters are consider like temperature , injection pressure , material of the mold ,analysis of mold flow etc.

This Paper contains Mold design consideration and design aspect of tooling and manufacturing.

REFERENCES

1. 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]
2. 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]
3. R. Dubay, B. Pramujati and J. Hernandez, “Cavity temperature control in plastic injection molding, IEEE July 2005 [911-916]
4. E O’Neill, C. Wilson and D. Brown, “The benefits of solid modelling in the plastics injection molding industry”, IEEE April 1997 [99-107]
5. M. Dastagiri , M. Prasad and M Annamacahrya , ”Application of Axiomatic Design in injection molding process,” IEEE 2010 [222-228]
6. Chi-Huang Lu and Ching-Chih Tsai,“Multivariable Self-Tuning Temperature Control for Plastic Injection Molding Process,“ [702-709]
7. C.G. Li, Yuguang Wu,“Evolutionary optimization of plastic injection mould cooling system layout design,“ IEEE 2010 [693-696]
8. Zhou Chun-ying, Wang Li-tao,“ Injection Mold Design based on Plastic Advisor Analysis Software in Pro/E,“ IEEE 2011 [205-208]
9. B. Bona, L. Giacomello, C. Greco, A. Malandra,“Position Control of a Plastic Injection Moulding via Feedback Linearization Machine“, Arizona December 1992 [2591-2593]
10. R. G. W. Pye,“Injection Mold Design an introduction and design manual for the thermoplastic industry“.
11. http://www.custompartnet.com/wu/InjectionMolding