Simulation And Design Of Injection Mould Die For Seven Segmental Gear

Guttula Hari Krishna[1], V Vijay Kumar[2]
[1]Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[1, 2] A.S.R Institute of Engineering and Technology

Abstract- *Injection moulding is the one of the most common method used in producing plastic products for many purposes ranging from a daily product to highly specified product. Presently the manufacturable Component is made with POM (DELRIN). The main aim of this project is Design and analysis of Injection Mould for SEVEN SEGMENTAL GEAR. The solid model of the segmental gear is designed in Nx-10. A Multi cavity die is proposed to satisfy the customer order. In the mould design the theoretical calculations economical no. of cavities, shot weight, clamping force, plasticizing capacity were calculated and type of Runner and Gate system was also decided. The materials are selected for the mould die parts and cost estimation for die manufacturing was also done. By using mould flow simulation software, the shot volume of 12.43 POM materials was filled into the multi-cavity impression at different melt temperatures ranging from 240oC to 270oC and in the analysis it was observed that for 270oC of melt temperature the quality is high. The cycle time is estimated as 17sec and the Injection Pressure is 27.69MPa. Through simulation results graphs were also plotted. In the mould design the economical no. of cavities, shot weight, clamping force were calculated and type of Runner and Gate system was also decided. The materials are selected for the mould die parts and cost estimation for die manufacturing was also done.*

Keywords- Simulation, Injection

I. INTRODUCTION

About Injection Molding Process: Injection molding process is one of the most versatile production methods in the plastic manufacturing industry. It is the process that is capable of producing molded parts of relatively intricate configurations with good dimensional accuracy. Injection molding is a method of forming or molding objects from powdered or granular type of Thermoplastic material. Thermosetting powdered materials are also being injection molded.

The powdered or granular plastic is fed from hopper to a heated chamber where it is heated and softened in a flow able condition and is then injected into a cool metal mould by a ram or screw ram. Pressure is held on molten plastic material until the mass has hardened enough for removal from the mould. Injection molding machines accommodates a cloth hopper, associate degree injection ram or screw kind plunger, and a heating unit. They are also known as presses, they hold the moulds in which the components are shaped. Pressure are rated by tonnage, which expresses the amount of clamping force that the machine can exert. This force keeps the mould closed throughout the injection method. Tonnage can vary from less 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations. The total clamp force required is set by the projected space of the half being moulded. This projected area is multiplied by a clamp force of from 2 to 8 tons for each square inch of projected areas. As a rule of thumb four or five tons/in is used for many product. If the plastic material is very stiff, it will require more injection pressure to fill the mould, thus more clamp tonnage to hold the mould closed. The required force may also be determined by the fabric used and therefore the size of the half, larger elements need higher clamping force.

Basic classifications of Mould based on construction:

Standard Moulds: The standard mould is that the most straightforward style, primarily the quality moulds is same as 2 plate molds construction, they divided in two side: cavity facet and core facet, cavity facet is that the facet that construct to flowing plastic material from nozzle to cavity components, basically they consists of tropical sprue, runner, core facet construct to create form for core, de-molding system and ejection system, at this facet we have a tendency to style ejection system Standard mould have one parting line, and have one opening direction. This type of mould use in all kinds of plastic parts that doesn't have undercut inner and outer screw. Light brown colorlittle and straight in ejection system is shown ejection pin.

Slider Mould: Development from this mould sort is that the used slider components in varied moulds varieties, basic slide mould is transfer horizontal movement of mould to vertical

movement, this sort of moulds is employed to form components with undercut.

Three Plate Mould: Basically three plate moulds has two parting line, and floating plate, floating plate support by support pin, since the mould has two parting planes, the runner system is placed on one facet of floating plate or build special plate that attach in floating plate, we tend to referred to as runner plate.

Split Cavity Mould: Basically the split cavity is same as 2 plate standard mold, but the cavity have split block to make undercut product or external threads. This type of mould use when the slider is not enough to make the undercut or the threads. The disadvantage of this mould are when use for little parts, the construction will difficult.

Mould with screw device: The mould special to make thread forming, the core can be rotate when demoldingprocess, both internal or external threads can be forming by this type mould.

Stripper Ejection Mould: system is placed on one aspect of floating plate or build special plate that attach in floating plate, we tend to tend to mentioned as runner plate. Moulds with complex and difficult design can be integrated or combined one to the others mould type.

Salient Features of Autodesk Simulation Mould Flow Adviser: Mould flow, 3D solids-based plastics flow simulation that permits plastics half styleers to see the manufacturability of their elements throughout the preliminary design stages and avoid potential downstream issues, which might cause delays and price overruns. “Design Plastic elements and Injection Moulds with Confidence” Discover, communicate, and resolve potential producing defects earlier within the development cycle with Autodesk Simulation Mould flow authority code. Today, manufacturing processes often include the production of injection-moulded plastic parts, a complicated process that can lead to unexpected delays and increased costs. More designers than ever are seeking the ability to confirm the manufacturability of plastic parts and avoid potential defects. Autodesk Simulation Mould flow Adviser software helps to simplify plastic injectionmolding simulation and optimize mould features such as gates, runners and cavity layouts.Intuitive to find out and use, it guides designers, mould manufacturers and engineers through simulation setup and results interpretation, providing recommendation on potential defects and suggestions for corrective action. Learn how changes to wall thickness, gate location, material, geometry and mould designs affect manufacturability. By helping to resolve and clearly communicate potential problems, Autodesk Simulation Mould flow Adviser software enables engineers to design plastic parts confidently.

Advantages of Using Autodesk Simulation Mould Flow Adviser:

- Validate part designs
- Optimize mould designs
- Guided results to improve manufacturability
- Reduce cycle times
- Communicate potential manufacturing defects
- Predict and correct part defects
- Simulates the most advanced moulding processes

Injection Molding Machine Components:

Barrel: The plastic material can enter the barrel once it's fed to the feed hopper. The barrel consists of cooling water channel, heater bands, screw and thermocouple whose function is to note the temperature in each section of the barrel. The time it takes for the plastic material room entering the barrel to the nozzle is called the residence time.

Heat Control: In the injection molding machine, the heats are produced by electric heater bands that surround the barrel. the machine requires a specific size and power output electric heater in order to maintain the stability and success of heating process. It is also one of the tricky things about replacing the heaters when one of them is no longer functioning.In most cases it'll be simple to exchange the heater band at intervals a similar size that matches in to the machine, in spite of having totally different power output. This action may lead to the instability of the heating method and method management issues later. Hence the thermocouples or RTDs ar unremarkably accustomed management and notice the barrel temperature that may maintain the right power output.

Nozzle: The nozzle is located in the end of the barrel. It provides the suggests that by that the soften will leave the barrel and enter the mould. It is also a region where the melt can be heated both by friction and conduction from a heater band before entering the relatively cold channel in the mould. Contact with the mould causes heat transfer from the nozzle and in cases where it is excessive it is advisable to withdraw the nozzle from the mould during the screw-back part in the molding cycle. Otherwise the plastic may freeze-off in the nozzle. A steel insert in AN injection mould that contain the upset hole and features a seat for the injection cylinder nozzle. The injection unit of the nozzle is located against the sprue bushing in order to convey the material without any leakage. Occasionally there may be a haul once the nozzle and also the

disorder don't match dimensionally, which might cause the soften plastic to ineffective to flow properly into the mould. The figures below show the visualization of the problem

Screw: The screw utilized in the currently a days injection moulding machine is what offers the potency and ease of the tactic. The injection screw is ready to perform many tasks within the moulding cycle. whereas it rotates ,it is able to soften, pressurizes,and conveys the plastic material from the fanny of the barrel to thr tip of the nozzle. And whilst the screw is not rotating .It is also capable of moving like a plunger to perform the injection of the molten plastic into the mould. The screw in injection moulding is commonly designed that its size increases gradually to let better mixing, melting and homogenization of the plastic materials. The srew design is divided into three parts. The first part is feed section where the material will go after entering the barrel. In the feed section most of the material will still be pellets. Then as the screw rotates they are conveyed to the next section which is the transition section. In the transition section, the solid pellets will start to be melting down and pressurized. Once all the plastic material is successfully molten, they will be conveyed to the third section which is metering section. In here the molten material will be accurately deposited with the approximate viscosity and temperature to do the injection.

Clamping System: Clamping system is critical to possess a alone purpose, that is to stay the mould tightly closed underneath spare pressure to let the liquified plastic fill within the cavity while not unseaworthy throughout the injection method. The system to try and do the specified clamping may use either a hydraulic ,mechanical(toggle),or both. The toggle clamping system clamps the mould employing a ratio developed through series of linkages. As the linkages are forced into a straight or closed position by the action of a hydraulic cylinder on a crosshead, the tie bars strain or stretch, and clamping forces are developed. The advantages of the toggle system are their fast motion, low oil flow requirement, and positive clamping action with no pressure loss. The disadvantage is is they allow the processor little or no control over tonnage variation, and frequent maintenance is necessary

General Methodology of Mould Design:

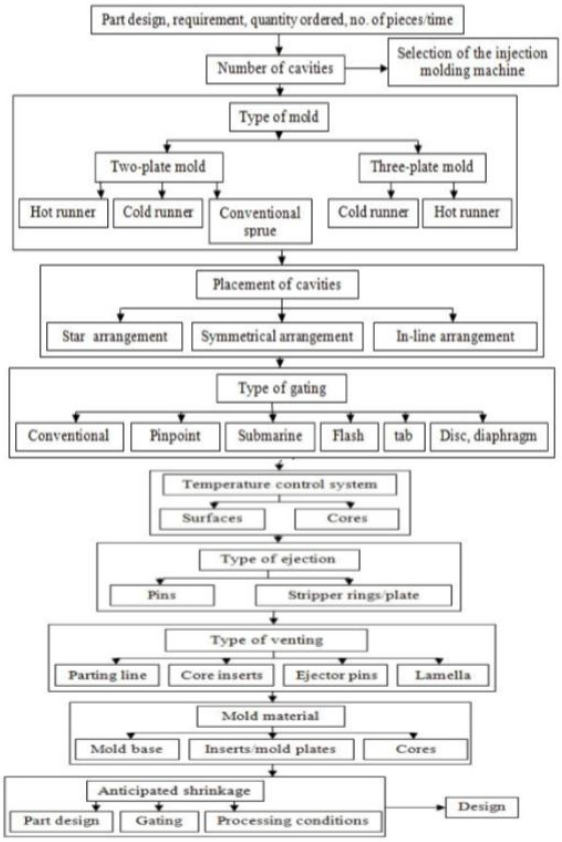

Fig: Methodology of module design

II. IDENTIFY, RESEARCH AND COLLECT IDEA

Ashby.M.F (2009): Plastic injection molding is one of the most important polymer processing operations in the plastic industry today. Ashby.M.F (2009) has stated that the plastic industry that is injection molding and involving manufacturing has high growth potential caused by the products are made nowadays is from the polymer material. The most establish method for producing plastic or polymer parts in mass production is plastic injection molding. This is a extremely efficient, precise and competent producing methodology, which may be automatic.

Framingham(2004): Autodesk Mould flow Plastic Insight is an in-depth process simulation tools to predict and eliminate potential manufacturing problems and optimize part design, mould design and the molding process itself. There are over nineteen distinct modules that can be used to simulate nine unique molding processes, by which thermoplastic injection molding is one of them.

Bryce.M.D (1996): Injection molding is a process of forming a product by forcing molten plastic material under pressure into a mould where it is cooled, solidified and subsequently released by opening the two or three halves of the mould.

Bryce.M.D (1996) has stated that the injection moulding is used for the formation of intricate plastic parts with excellent dimensional accuracy. A large variety of things related to our standard of living square measure made by approach of injection moulding. Typical product classes embody house wares, toys, automotive elements, furniture, rigid packaging things, appliances and medical disposable syringes.

Ching-Chih and Chi-Huang Lu (1998): Chi-Huang LU,(1998)Findings-The extruder typically consists of a large barrel divided into several constant temperature zones with a hopper at one end and a die at the other Polymer is fed into the barrel in raw and solid particle form from the hopper and is pushed forward by a powerful screw.While passing through the temperature zones with bit by bit increasing temperature, the raw chemical compound is bit by bit heated. The heat made by the warmers within the barrel, at the side of the warmth free from the friction between the raw chemical compound and therefore the surfaces of the barrel and screw, causes the melting of the feed chemical compound, that is then pushed by the screw into the molding mechanism from the die. Generally speaking, the quality of the extrude depends upon the uniformity of temperature distribution, magnitude of the temperature in the barrel, back pressure, and the homogeneity of the physical mixing.

III. IMPROVEMENT AS PER REVIEWER COMMENTS

Instead of water different types of oils may be used for effective and uniform cooling for decreasing cooling time and increasing production rate. Thermal and stress analysis on whole die may be performed by using analysis software's. Automation of the process by using part handling and inspection. 8 cavities can also be designed and manufactured such that a shift can be reduced and production rate is increased.

IV. RESULTS

The 7 segmental gear was modeled in nx-10 as per the given dimensions. The part was imported into Autodesk Mould Flow Simulation Software and the best gate location was decided near to the bottom of the part. For varying melt temperatures ranging from 240 oc to 270 oc the multi cavity impression was filled and analysed. From the simulation it is concluded that the quality of components is high at 270 oc. For the preferred 270 oc melt temperature the Injection Pressure is 2.044tons/sq.cm and the cycle time is estimated as 17 sec. For filling the multi cavity impression the shot weight of 17.658 gms is injected. Circular type of runner used with Submerged Gate. The clamping force is obtained as 14.5218 Tons. Through Simulation it is concluded and observed in the graphs that as the Temperature is increasing the Fill Time, Injection Pressure, Pressure Drop is decreasing where as Flow Front Temperature and Quality is Increasing. The Finger Cam Actuated Multi Cavity Injection Mould of "7 SEGMENTAL GEAR" has been successfully designed and manufactured as per customer requirement of 26 lakhs per annum. Profit on the Tool is Rs. 27,074 and total profit along with production of components is Rs.35,12,754.

V. ACKNOWLEDGMENT

I would like to show my gratitude to my guide prof., who helped and support me for this project. Most of the data in this paper is based on the literature review of my MTech project.

REFERENCES

[1] R.G.W.PYE, "Injection mould design"
[2] CITD Data book of Tool Engineering Parameters
[3] http://en.wikipedia.org/wiki/Injection_molding.html
[4] "Mould master data hand book", Company standards book, 2003.
[5] Crawford.R, "Plastic engineering", 3rd edition, Oxford: Butter worth - Heine-Mann, 1998.
[6] Edward.A.Mucio, "Plastic processing technology", ASM International, 1994.
[7] D.V.Rosato&M.G.Springer "Concise encyclopedia of Plastics", Kluwer Academic Publishers, 2000.
[8] Peter Kennedy, RongZheng, Flow Analysis of Injection Molds, 2nd Ed., Hanser Publications, Cincinnati, U.S.A, 2013
[9] Harry Pruner and Wolfgang Nesch, "Understanding Injection Molds", Hanser
[10] "Autodesk Simulation Mould Flow Adviser 2016", Proe-5.
[11] "HASCO", Standards Catalogue, 2010