International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 04 | Apr 2019 www.irjet.net p-ISSN: 2395-0072

Simulation for Optimum Gate Location In Plastic Injection Moulding for Spanner

Atul B. Munankar[1], Niranjan A. Sitap[2], Alok A. Tembhurkar[3], Mahesh B. Thakur[4], Sandip S. Kanse[5]

[1,2,3,4]*Student, Mechanical Engineering, BVCOE NaviMumbai, India*
[5]*Associate Professor, Mechanical Engineering, BVCOE, Navi Mumbai, India*

Abstract - *As there is tremendous demand of various plastic products, plastic producing industries are increasing in a fastest rate. Injection moulding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide style of product area unit factory-made exploitation injection moulding, that vary greatly in their size, complexity, and application. The optimum gate location is one of the most important criterions in mould design. Moldflow analysis is effective simulation tool to optimize the gate location at lowest possible cost with least defects and it requires less time to achieve a quality result with material waste as compared with typical trial error methodology on production floor. In this project an analysis has been performed by taking varying gate locations for a spanner as a component . Autodesk Moldflow Advisor simulation software from Autodesk was used for the analysis and optimum gate location was found with least defects.*

Key Words: Injection moulding; Autodesk Moldflow Advisor; Simulation; Gate Location; moulding defects

1. INTRODUCTION

One of the most common methods of converting plastics from the raw material form to an article of use is known as injection molding. This method is most –usually used for thermoplastic materials which can be in turn dissolved, reshaped and cooled. Injection mould components are a feature of almost every functional manufactured article in the modern world, from automotive products through to food packaging. This versatile method permits United States to provide top quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology over the last fifty years close to.

To understand the engineering and operation of modern day injection molding machines, it is useful to first look at the not too distant origins of the process. The first injection molding machines were primarily based around pressure die casting technology used for metals process, with patents registered in the USA in the 1870's specifically for celluloid processing. Further major industrial developments failed to occur till the 1920's once a series of hand operated machines were created in Federal Republic of Germany to method thermoplastic materials. A simple lever arrangement was wont to clamp a two piece mould along. Molten plastic was then injected into the mould to produce the

mould component. Being an inherently low method, it was limited in use. Pneumatic cylinders were accessorial to the machine style to shut the mould, although little improvement was made. Hydraulic systems were first applied to injection molding machinery in the late 1930's as a wider range of materials became available, although the machine design was still largely related to die casting technology.

Large-scale development of injection molding machinery design towards the machines we know today did not occur until the 1950's in Germany. Earlier machines were supported a straight forward plunger arrangement to force the fabric into the mould, although these machines soon became inadequate as materials became more advanced and processing requirements became more complex. The main drawback with a simple plunger arrangement was that no soften mix or blend can be without delay imparted to the thermoplastic material. This was exacerbated by the poor heat transfer properties of a chemical compound material. One of the foremost necessary developments in machine style to beat this drawback, which still applies to modern processing equipment today, was the introduction to the injection barrel of a plunging helical screw arrangement. The machine subsequently became known as a 'Reciprocating Screw' injection molding machine.

2. LITERATURE REVIEW

Plastic injection moulding, the polymer analogue of die casting for metals, is the most widely used technique for fabricating thermoplastic materials. It is a high rate production process, with good dimensional control. Moreover, this is a net-shape process, so highly complex components can be produced in the finished state. Plastic injection moulding (PIM) has many advantages such as short product cycles, high quality part surfaces, good mechanical properties, low cost, and light weight, so it is becoming increasingly more significant in today's plastic production industries. Costs can be reduced further by the integration of components, while there is the potential for weight savings over metal counterparts. A finely pelletized or granulized plastic is forced, at an elevated temperature and by pressure, to flow into, fill and absorb the shape of a mould cavity. Ming Zhai& Ying Xieexplained [6]that gate location optimization can be done by changing the gate location. He concluded that the gate location optimization can be done to achieve a balanced flow for injection molding. Yung-Kang. Napsiah Ismail & A.M.S. Hamouda. This paper presents the design of plastic injection mould for producing a product. The

plastic half was designed in numerous sorts of product, but in the same usage S.R. Pattnaik [1]studied gate location optimization by selecting two gate locations for L-shaped plastic part, Moldflow plastic adviser software is used for simulation. A comparative analysis for locating the optimum gate location is also performed with the assistance of mould flow software and selected the optimum gate location. Jagannatha Rao M B, Dr. Ramni [4] analyzed plastic flow of two plate multi cavity injection mould for plastic component pump seal, CAD/CAE technology facilitates the design of mould. He allover simulation reduces range and complexness of manual setup operations.

R.A. Tatara [5]studied the injection molding processing for dispensing closure having an integral hinge with minimal input; and concluded that Moldflow is able to perform a relatively sophisticated analysis of an injection molding function. One half is victimization clip perform and another half is victimization stick perform. In the laptop power assisted design (CAD), two plastic parts were drawn in 3dimension(3D)view by using Pro-Engineer(Pro/E) parametric software .In the computer-aided manufacturing(CAM), Pro Manufacturing from Pro/E parametric software was used to develop the machining program. For mould style, the product was designed into two changeable inserts to produce two different types of plastic production one mould base. Before continuing to injection machine and mould style, this part was analyzed and simulated by using Part Advisor software. From the analysis and simulation we are able to outline the foremost appropriate injection location, material temperature and pressure for injection. The predicted weld lines and air trap were also found and analyze .OYETUNJI .A development of small injection molding machine for forming small plastic articles in small-scale industries was studied. This work which entailed design, construction and test small injection molding machine that was capable of forming small plastic articles by injecting molten resins into a closed, cooled mould, where it solidifies to administer to specified merchandize was developed. The machine was designed and created to figure as a model for manufacturing terribly tiny plastic parts.

3. METHODOLOGY

3.1. Definition of Optimization Problem

To define the problem, we must first identify all the inputs and outputs of the system in PIM. Optimization of gate location is the main criterion in the mould tool design. The production of injection moulded parts is a complex process where, without the right combination of material, part and mould design and processing parameters, a multitude of manufacturing defects can occur, thus incurring high costs. The injection moulding process itself is a complex mix of time, temperature and pressure variables with a multitude of manufacturing defects that can occur without the right combination of processing parameters and design components. Determining best initial process method parameter settings critically influences productivity, quality, and prises of production within plastic injection molding (PIM) business.

3. Study of Component (Spanner)

The component for which mould gate location is to be selected is "Spanner". This spanner is of material Glassfibre 66 Nylon which is most commonly used in Autodesk Moldflow Advisor 2016. A spanner is used for turning a nut, bolt or similar fixing that is done to tighten. The spanner is used to grip the given fixing (whether it is nut, bolt, concrete screws etc.)and turrn it, allowing you to apply torque and tighten the nut onto the bolt. During the study of component following points were found important to be considered to find the gate location in mould design of spanner as shown in fig.3.1.

1. The component is thick and chunky and is appropriate for 3D modelling analysis
2. The component is design in such a manner so that it can turn or fix the nut or bolt on required operation.
3. The spanner is of size of bolt diameter on one side ϕ17 and ϕ13 on other respectively..

Fig 3.1 model of spanner designed in CATIA V5

3.3 Parameters for Finding Gate Location

Proper gate location of a component depends on different parameters such as melt temperature type of material and its properties, type of gate, number of gate locations etc

Material Type: Glassfibre Reinforcements: Fibreglass Reinforced Plastics, FRP, is an excellent choice of material for the construction of chemical storage tanks, piping systems, apparatus and other types of industrial process equipments. The FRP material properties may beat many conventional materials, such as steel when it comes to chemical and corrosion resistance. Little maintenance and long product life time, that is what well engineered FRP equipment promise. Plastic material used for spanner is glassibre 66 nylon, so parameters are as follows.

Grade : Nylon 66 (RTP 0207)

Melt temperature : 290°c.

Number of Gate Locations:

Gate location for a component can be selected more than one at a time, as per the requirement/ But for our component we will use only one gate location because our component is small. If the number of gate locations are increased then it will give more weld lines. So it is good to avoid extra gate locations in the component. For finding optimum gate location, the component is checked with two different gate locations. These locations are as follows.

1. Gate 1:- At the centre of the spanner handle
2. Gate 2:- At the centre of curved surface of spanner on the side of ϕ13

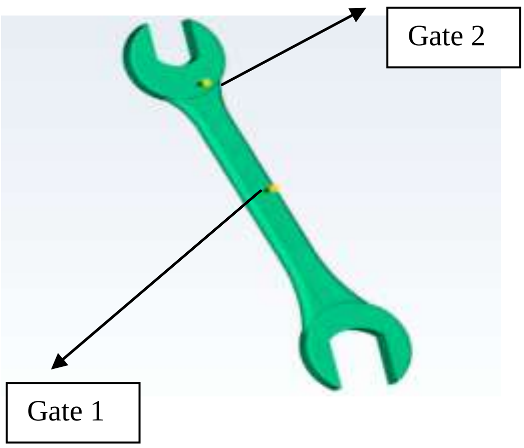

Fig3.2. gate locations selected in spanner

Now with these two gate location, as shown in fig.3.2, simulation is carried out of the component, and from results the best gate location is obtained.

4. Flow simulation of spanner component

The benefits of using CAE software to design and engineering components and feed system include.

1. Innovative, reliable and high quality products and processes.
2. Fewer ROI, due to fewer physical prototypes and test setups.
3. A more flexibl0e and responsive information based development process enabling modification of designs at later stages.
4. A seamless exchange working of data , regardless of location, industry, CAD environment etc.
5. Lower manufacturing cycle times..
6. Improved and consistent component quality.

The simulation of selected component is carried out by using "Autodesk Mouldflow Advisor". The resolution of component is carried out in high analysis resolution form. The simulation gives the following results for fill time, Quality predictions, injection---pressure, Air traps and Weld lines.

4.1. Fill Time

One of the important process parameters to establish and record for any injection moulded part is its injection or filled time. The fill time result shows the position of the flow front at regular intervals as the cavity of the component fills. All the regions which have same colour are filled simultaneously. The result we get at the start of the injection is dark blue and the last remaining areas are to be filled with red colour. If the part which is short shot, the section which is not filled has no colour. The fill time can be used to estimate possible areas of short-shot, over packing, weld lines and air traps.

The fill time for the first gate location is 0.3054 sec. and for the second gate location is 0.5098 sec. The results as shown in the fig.4.1 and 4.2 gate location 1 requires less time to fill the cavity then the gate location 2.

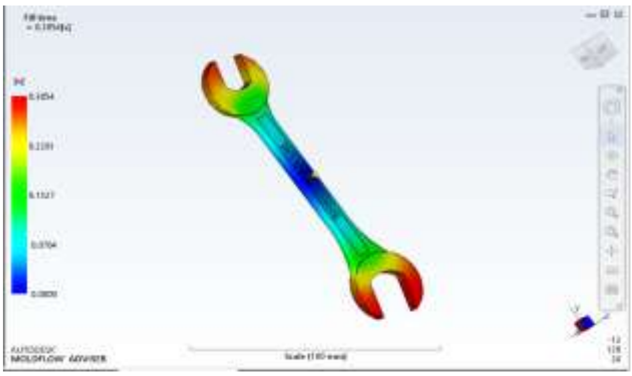

Fig4.1: Fill time of gate location 1

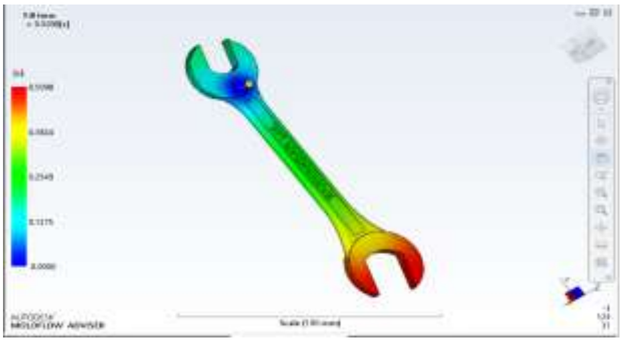

Fig.4.2:Fill time of gate location 2

4.2. Quality prediction

The Quality prediction result's won't to estimate the standards of the mechanical properties and look the half. This result is derived from pressure, temperature and other results. The colours displayed in the Quality prediction result indicate the following.

1. Green- Will have high quality.
2. Yellow- May have quality problems.
3. Red- Will definitely have quality problems.

The result as shown in fig.4.3 and 4.4 shows maximum percentage of high quality part of a component, less percentage of medium quality part of a component and no gate shows low quality part of a component. The quality predictions of both the gate location are given below.

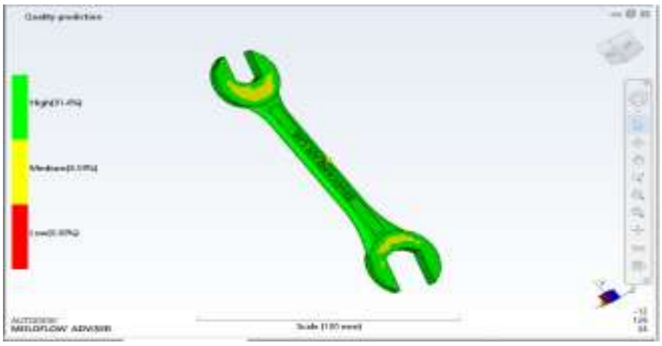

Fig 4.3 Quality prediction result of gate 1

Fig 4.4 Quality prediction result of gate 2

4.3 Time to reach ejection temperature

The time to reach ejection temperature results show the amount of time required to reach the ejection temperature, which is measured from start to fill.

If the half has not frozen by the tip of the cycle time provided, a projected time to freeze is displayed within result.

Ideally, the part should freeze uniformly. Areas of the half that take longer to freeze could indicate thicker areas of the half or areas of shear heat throughout filling and/or packing.

If along period of time to reach the ejection temperature is caused by thick areas in the part, consider redesigning the part. Long periods of time that time that are due to shear may be difficult to solve. Reducing the shear may cause the time to reach ejection temperature to adversely effect volumetric shrinkage and warpage.

If the resulting values of the part as whole appear high, action to reduce the cycle time may need to be taken, Such as reducing the mould and melt temperature.

The results as shown in fig. 4.5 and 4.6 shows that gate location one takes 22.79 sec. and gate location two takes 22.97 sec. which is more in gate location two .So, gate location one is more optimum for time to reach ejection temperature.

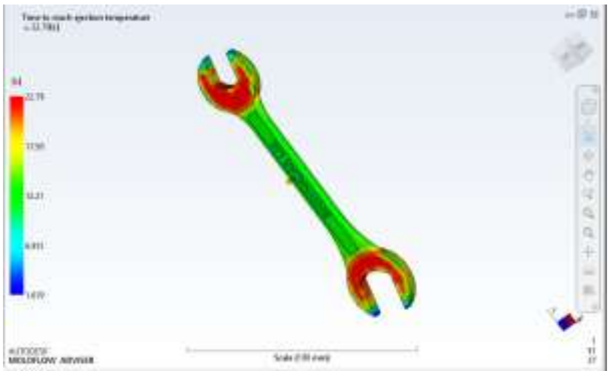

Fig.4.5 time to reach ejection temperature at gate 1

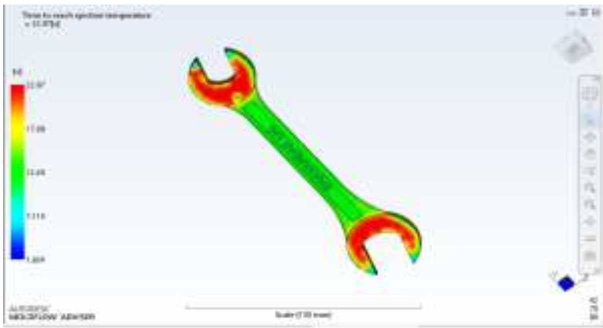

Fig.4.6 time to reach ejection temperature at gate 2

4.4 Air traps

An air traps is the air that is caught inside the mould cavity. It becomes cornered by convergence compound soften fronts or as a result of it didn't shaped the mould vents, or mould inserts, which also acts as

vents. Air traps locations are usually areas that fill last. Lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps and the resulting defects.

Problem caused by air traps:

1. Racetrack effect: The racetrack effect occurs when molten plastic flows into thicker regions more easily than thin regions. The flow divides then fills thicker sections before combining again to fill the diluents sections. The recombined flow will reverse to satisfy the oncoming flow within the diluent section.
1. Hesitation: In a part with multiple flow paths, the flow can slow down or hesitate in thin regions.
2. Unbalanced flow paths: Flow paths do not needs to exhibit the racetrack effects or hesitations to have unbalanced flow. In a give up uniform thickness, the physical length of flow paths may vary, and again, air traps may occur.
3. Inadequate venting: Lack of vents or undersized vents in these last-to-fill areas are common cause of air traps.

Remedies:

1. Balance flow paths.
2. Avoid hesitation and racetrack effects.
3. Balance runners. Changing the runner system can alter the filling patterns in such a way that last-to-fill areas are located at the proper venting locations.
4. Vent appropriately. If air traps do exist, they should be positioned in regions that can be easily vented or ejection and / or vent pins added so that air can be removed.

4.4 Injection Pressure

The injection pressure result, which is produce by Fill analysis, shows the maximum injection pressure value obtained before the velocity / pressure switch-over occurs during the filling phase. The pressure at specific location starts to increasing only after the melt front reaches that location. The pressure continues to increase as the melt front moves past, due to increase in flow length between these specific location and the melt front. The magnitude of the pressure or pressure gradient depend on the resistance of the polymer of the mould. This is a result of compound with high viscousness needs a lot of pressure to fill the cavity.

The pressure result as shown in fig. 4.7 and 4.8 shows that the first gate location require minimum injection pressure as compared to second gate location. The pressure result shows that the first gate location require the pressure of 8.744 MPa and for second gate location is 10.17 MPa. Higher injection pressure indicates the occurrence of higher shear rate and shear stress level.

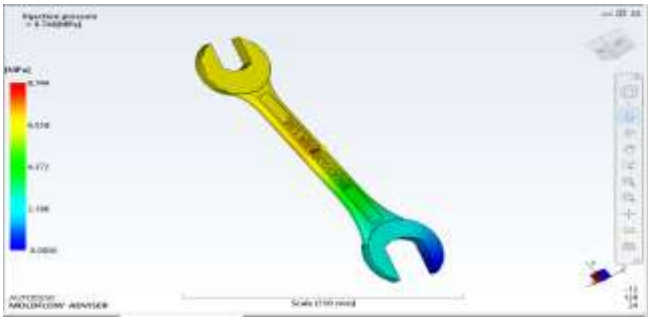

Fig 4.7 Injection pressure at gate location 1

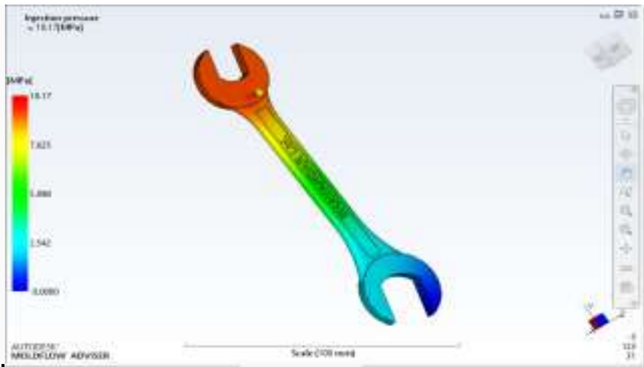

Fig 4.8 Injection pressure at gate location 2

5. Simulation results summary

The component is simulated on the Autodesk Moldflow Advisor 2016 Software. The result obtained from the stimulation helps to identify the correct gate location and helps to identify the possible defects in the design phase itself, which may arise due to actual production. Stimulation result obtained are summerised in the table below.

Table 1: Results summary table

Results	1st location	2nd location
1. Fill time	0.3054 sec	0.5098 sec
2.Quality Predictions	High, Medium	High, Medium

3.Time to reach ejection Temperature	22.79 sec	22.97 sec
4.Air Traps	Present at top	Present at top
5.Injection Pressure (MPa)	8.744	10.17

Table 2: Reference of colour

	Most adoptable values. (compared to other values)
	Values that can be adoptable at some extent.(compared to other values)
	Values which are not preferable to adopt. (compared to other values)
	Values which are not adoptable at any cost. (compared to other values)

From the above simulation results of the component it can be seen that the 1st gate location suggests itself "most adaptable values". The 2nd gate location shows the values that are adaptable at some extent. Therefore the 1st gate location can be selected as final gate location for further process of mould design. So gate can placed "At centre of the handle" of the component (gate location).

6. Conclusions

In this study, the spanner component is flow simulated for two gate locations to find the better location. The following conclusions can be derived from the above stimulation.

- The fill time required for gate location 2 is more than gate location 1, so fill time of gate location is considered as most adoptable value.
- The minimum pressure is required for first gate location that is 8.744 MPa, which gives the occurrence of minimum shear rate and shear stress level.
- The time to reach ejection temperatures is maximum in gate location two, which gives thicker areas of the part of shear hitting during
- filling.. Therefore gate location one is most preferable.
- All cases may form small amount of air traps, which can be avoided by providing proper air venting.

So finally by combining all results of the gate locations of component, first gate location is best for further mould design process.

REFERENCES

1. S.R. Pattnaik, D.B.Karunakar, P.K. Jha. "Application of computer simulation for finding optimum gate location in plastic injection Moulding process", journal IJIR, vol-3, issue-5 2017
2. Y. P. Tidke, A.V. Dhote, Dr. R. C. Patil. "Study of optimization in PIM using Taguchi method." , journal IJRASET, vol-5, issue-4, April 2017
3. Sanjay k. Nayak, Pratap Chandra Padhi, Y. Hidayathullah. "Fundamentals of Plastics Mould Design", Tata McGraw Hill Education Private Limited, 2012.
4. Jagannatha Rao M B, Dr. Ramni, "Analysis of Plastic Flow in Two Plate Multi Cavity Injection Mould for Plastic element for Pump Seal", International Journal of Scientific and Research Publications, Volume 3, Issue 8,ISSN 2250-3153 August 2013
5. R.A. Tatara, R. M. Sievers, Valentin Hierzer, "Modeling the injection moulding processing of a polypropylene closure having an integral hinge", Journal of Materials Processing Technology
6. Ming Zhai& Ying Xie, "A study of gate location optimization of plastic injection MOULDING using sequential linear programming", Int J AdvManufTechnol (2010) 49:97–103.
7. Zhou Chun-ying, Wang Li-tao," InjectionMold Design based on Plastic Injection mould cooling system layout design," IEEE 2010 [205-208]

BIOGRAPHIES

Atul B Munankar
"Student, Bharati Vidyapeeth college of engineering, Navi Mumbai"

Niranjan A. sitap
"Student, Bharati Vidyapeeth college of engineering, Navi Mumbai"

Alok A. Tembhurkar
"Student, Bharati Vidyapeeth college of engineering, Navi Mumbai"

Mahesh B. Thakur
"Student, Bharati Vidyapeeth college of engineering, Navi Mumbai"

Prof. Sandip S. Kanase
"Associate Professor, Bharati Vidyapeeth college of engineering, Navi Mumbai"