International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 3, Vol. 2, Sept 2015.
e-ISSN: 2394-5141, p-ISSN: 2394-5133, pp 122-129

Design and Analysis of Injection Mould for Hydrogol Dripper

Manjusha Narayane
manjusha.narayane@gmail.com

Department of Mechanical Engineering,
St. Vincent Pallotti College of Engineering & Technology,
Nagpur, India

***Abstract*— In this project, a two plate injection mould design and mould flow analysis of 12-cavity injection mould for a given component was taken according to the customer requirement. Present mold 12-cavity injection mold for Irrigation valve found a defect as short filling. This projects aims to design and analysis of mould and proper correction of mould. Material selected for Plastic component "Irrigation Valve (Hydrogol Dripper)" was HDPE (High Density Polyethylene) . The 3-D model of the component and extraction of core and cavities was performed in Unigraphics (NX6) software. Drafting and detailing made on AutoCAD software. Auto Desk Mould Flow Analysis software is a powerful Analysis tool to analysis and predict the defects in the component.**

Index Terms—Injection Mould, Mold

I. INTRODUCTION

The injection molding has seen steady growth since its beginnings in the late 1800's. The technique has evolved from the production of the simple things like combs and buttons to major consumer, industrial, medical, and aerospace products.

The invention of an injection molding machine was achieved by John Wesley who injected hot celluloid into a mold which resulted in billiard balls which were used as a replacement for ivory which was based on the pressure die casting technique for metals.

The main concept of plastic molding is placing a polymer in a molten state into the mold cavity So that the polymer can take the required shape with the help of varying temperature and pressure. There are different ways of molding a plastic some of them are blow molding, Injection molding, rotational molding and compression molding. Each technique has their own advantages in the manufacturing of specific item.

This days mould and product designing involves the use of computer designing software's such as AutoCAD, PRO-E, UG (NX), Solid Edge and SolidWorks for the part designing and assembly software for the aggregate designing of the mould assembly making the mould manufacturing precise and accurate for the injection moulding manufacturing process of plastics and enabling wide varieties of tolerance with production varieties.

II. LITERATURE SURVEY

Injection molding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost. Faced with global competition in injection molding industry, using the trial-and- error approach to determine the process parameters for injection molding is no longer good enough. Factors that affect the quality of a molded part can be classified into four categories: part design, mold design, machine performance and processing conditions. The part and mold design are assumed as established and fixed. During production, quality characteristics may deviate due to drifting or shifting of processing conditions caused by machine wear, environmental change or operator fatigue. Determining optimal process parameter settings critically influences productivity, quality, and cost of production in the plastic injection molding (PIM) industry. Previously, production engineers used either trial-and-error method or Taguchi's parameter design method to determine optimal process parameter settings for PIM. However, these methods are unsuitable in present PIM because of the increasing complexity of product design and the requirement of multi-response quality characteristics. This article aims to review the recent research in designing and determining process parameters of injection molding. A number of research works based on various approaches have been performed in the domain of the parameter setting for injection molding. These approaches, including mathematical models, Taguchi method, Artificial Neural Networks (ANN), Fuzzy logic, Case Based Reasoning (CBR), Genetic Algorithms (GA), Finite Element Method (FEM), Non Linear Modeling, Response Surface Methodology, Linear Regression Analysis, Grey Rational Analysis and Principle Component Analysis (PCA) are described in this article. The strength and the weakness of individual approaches are discussed. It is then followed by conclusions and discussions of the potential research in determining process parameters for injection molding.[13]

The article reveals that Cessna Aircraft Co. contracted Schaumburg, Illinois-based PM Mold, an injection molder and mold builder, to produce interior plastic injection components for the Cessna Mustang aircraft. PM Mold General Manager Larry Hauck contacted CAE Services Corp. to perform the mold flow analysis of the parts. PM Mold exclusively use Moldflow's Plastics Insight software for in-depth part and mold design simulation.[15]

Injection moulding is one of the most important processes of plastics and rubber compounds processing. It enables the production of very complex plastics and rubber parts, even parts made of different plastics and rubber compounds,The system for injection moulding of polymers consists of different elements such as the mould, injection moulding machine and device for mould temperature regulation (tempering), and additional elements: dryers, robots, etc. The paper presents the analysis of energy consumption in injection moulding systems, as well as the possibilities of energy savings in injection moulding process, starting with material selection, in order to obtain a more energy efficient process.[16]

III. PROBLEM IDENTIFICATION

In present injection mold base found short shot problem. Short shots in injection molding typically appear in the last cavities in 12-cavity injection mold that you expect the plastic to fill. Usually that is in the cavity furthest from the sprue (in an unbalanced runner system). The molder expects any short shot will occur at that location because it represents the longest flow length for any of the cavities.

IV. METHODOLOGY

A. *Aims and Objective*

As an objective of this thesis project, after covering the literature of basic plastic material analysis and basics of mould designing theories for injection moulding, the Hydrogol Dripper product part design and analysis is done based on researches, functionality, consistency and general peculiarity keeping its own identities as a plastic product part design and the following objectives carried out:

- Learn to use CAD software to design a product and its mould
- Understanding the Mould Structure and its function especially in the cavity side of the mould.
- Learn to use mold flow analysis software for injection mold.
- To design and test the plastic injection mold for the specific product
- Using Mold Flow analysis software finding out analysis result.

B. *Proposed Methodology*

This thesis presents a practical Tool Design procedure/methodology of an injection mould. In this Part the main details that are discussed are the appropriate designing methods to accomplish the mold design for the required product.

The appropriate steps in designing and analysis of a mold are

Product Design**:** As per customer requirement to produce Hydrogol Dripper plastic component by injection moulding process. Mould design and Product design has done in 3d Modellng tool Unigrahics (UG NX6)

Flow Chart: The flow chart of the methodology followed for the injection mould design is as follows

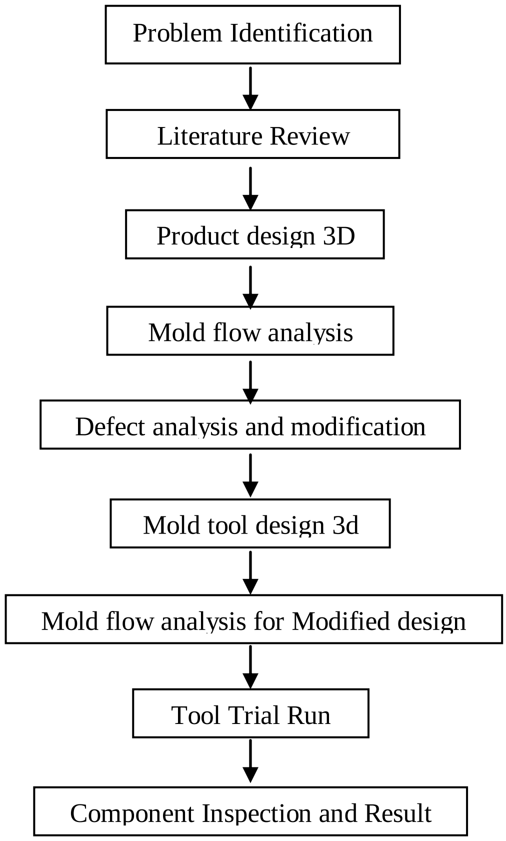


Figure 1: Flow chart of the methodology

C. *Software Assistance*

On meeting the required level of perfection, accuracy and proper product finishes now a days computer aided designing is vital and crucial. Especially when we talk about plastic product designing, the tie goes beyond apparent level of notice, simply explaining, it is almost impossible without the aid of this applications. As briefly mentioning some of the advantages:

- Creating a visual appearance of the part design in a professional discreet manner before production.
- Taking the Limit of dimensional consistency and accuracy almost to perfection level and creating the channel for editing, enhancing the part design before production.
- Achieving a design in a minimized time span with different editable and enhanced parameters.

There are broad categories of computer aided designing and simulation software, depending on the required result and feature of the product, they can be chosen and applied on computer based designing procedure accordingly.

AutoCAD, Unigraphics (NX6), and PRO-E: Applicable for 2D and 3D designing of a required part design with each a distinct parameters, Interface and further can be employed for assembling of the part design for a mould aggregate compilation.

Autodesk Moldflow Analysis : Analysis of a melt flow by setting injection point and optimizing it for optimal phase and values of filling the cavity, clamping force and also resulting with optimal analysis of filling time, under varying mould designing parameters such as clamp force, temperature and pressure it analyses the optimal phase of injection parameters.

V. Design and Analysis

A. Plastic Component Identification

Plastic Component name is Hydrogol dripper, it is cylindrical, designed with a long, wide labyrinth with high clogging resistance drippers due to a very large water passage and internal filtration area and two water outlets. This configuration provides a turbulent water flow which minimizes the formation of residues that may cause clogging.

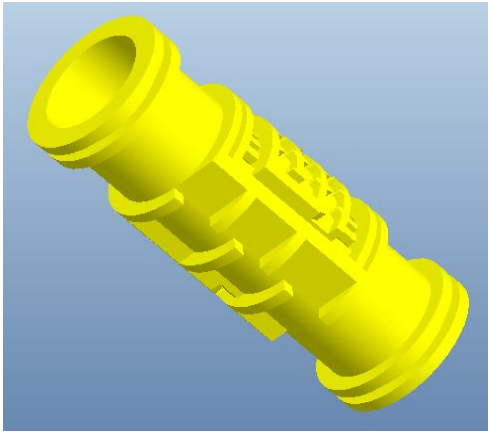


Fig.2 Plastic Component Hydrogol dripper

Material: HDPE (High Density Polyethylene) Flow rates are calculated at 1.0 bar [15 psi] pressure. The flow rate may vary according to the inlet pressure. Recommended working pressure range: 0.8 - 2.0 bar [12 - 30 psi]

Applications: for all types of field crops, also for fruit & orchards also for greenhouses. Hydrogol for small and large irrigation systems.

B. Defect Analysis and solution

In present 12-cavity injection mould for Hyrogoal Dripper found short filling defect in the last 2 cavities. In industrial processes, injection moulding produce flawed parts. In the field of injection moulding, troubleshooting is often performed by examining defective parts for specific defects and addressing these defects with the characteristics of the process itself. Trials are often performed before full production runs in an effort to predict defects and determine the appropriate specifications to use in the injection process.

When filling a new or unfamiliar mould for the first time, a technician/tool setter may perform a trial run before a full production run. He starts with a small shot weight and fills gradually until the mould is 95 to 99% full. As it determines cycle time and the quality and consistency of the product, which itself is an important issue in the economics of the production process. After the full cycle time run, last two cavities in injection molding not filled completely. Short shot type of defect found in 12 cavity injection mold.

In order to take countermeasures against short shots, it is necessary to verify which of the above two types the short shot belongs to

1) In the case of short shots caused by the solidification at the end of the flow

2) In the case of short shots due to air traps

- -Countermeasures related to molds
- -Countermeasures related to the injection molding conditions
- -Countermeasures related to the design of the molded product

In defect analysis, it is found that short shot caused by solidification at the end of flow. Pressure of molten plastic flow is not distributed uniformly to all the cavities. Cavity filling begins at the cavities nearest the sprue and last cavities not filled completely. In production trial, below all processing parameters checked to recover mould defect.

1. Changing the injection speed and changing the flow pattern
2. Changing the screw speed, and the pressure selection position.
3. Making the injection speed slower
4. Setting the cavity surface temperature higher.
5. Setting the mold clamping force a little lower.

In investigation of defect short shot, it is found that, In-line type runner is not balanced to all 12 cavities. Therefore it is required to analyze and modify design. There are two types of naturally balanced runner systems; symmetrical runner configurations, and circular runner configurations.

The 12-cavity injection mold has symmetrical in-line type runner. Modified runner cavity layout design as shown below. Each cavities are equally placed at 25mm. First cavity from the sprue is away from 52mm. Six cavities are placed both side of sprue bush so that pressure of plastic molten flow distribute equally to all cavities.

In an ideal design, all cavities would fill at the same instant and pressure, thus producing uniform, high quality parts. Thus runner design made to make pressure flow uniform to all cavities. S-type of sub-runner distribute same pressure to four cavities. And last two cavities are filled by liner flow of molten plastic. Following are the design consideration for selection of this type of runner layout as shown in figure 3.

1. Artificially balanced all 12 cavity.
2. Pressure velocity distributed same to all 12 cavities (shown on mold flow analysis)

Reason behind to select s-type runner layout is that to control the flow rate of molten plastic material. At the first cavities has more pressure and fills completely. Pressure distributed towards last cavities because of s-type runner layout.

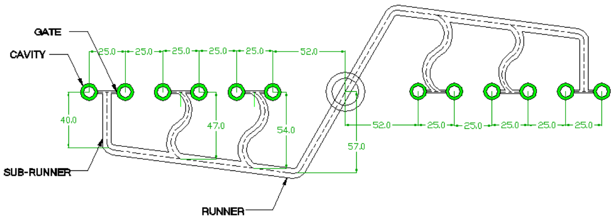


Figure3: Modified Runner layout

Long tapered cold runner are U-shaped of width 6.25mm and height 5.25mm,cold runner U-shaped of width 7mm and height 6mm, s-type cold runner of width 6mm and height 5mm, gate width 1.8mm and height 1mm as shown below figure 2.

C. Tool Design

1. Construction:

Split type cavity mold basically used for making undercut product or external threads. In this mold, the jaws form the mold cavity. The jaws can be inserted diagonally on the nozzle side and they then move on the diagonal to the outside when the mold is opened by means of a pull tab. Thus, the injection molded part is released for demolding.

Split mold design for Injection machine Electronica EMP 210 MPCL, Mould height minimum 200mm,max 600mm, Clamping Area 520X420mm, Shot Capacity 514 gm.

For structural dimension of plastic part and corresponding cavity structure, wherein plastic part are made from HDPE (High Density Polyethylene), shrinkage is 1.5-4%. The dimensions of core and cavity is to be calculated.

Figure 4 shows a schematic of the conceptual design of sectional front view of split 12 cavity injection mold.12 cavity split mold consists of below main parts: split block, finger cam, heel block, guide bush, guide pillar, ejector pin, ejector rod, ejector plate, cavity plate, core pin, sprue, runner plate etc.

From Figure 5 shows Top view of Assembly- Split 12 cavity Injection Mould (Moving Half). Top Plate 496x396mm, four guide bush and pillar at each corner of top plate is shown.

Section A-A cut through guide bush-pillar, ejector rod, core-cavity, finger cam. The sectional front view is shown in figure 5

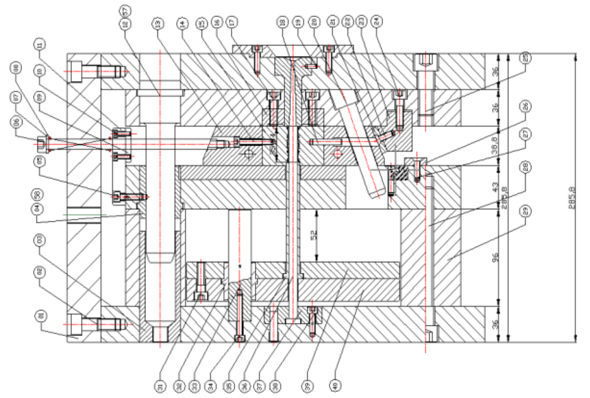


Figure. 4: Section view of Split 12 cavity Injection Mould

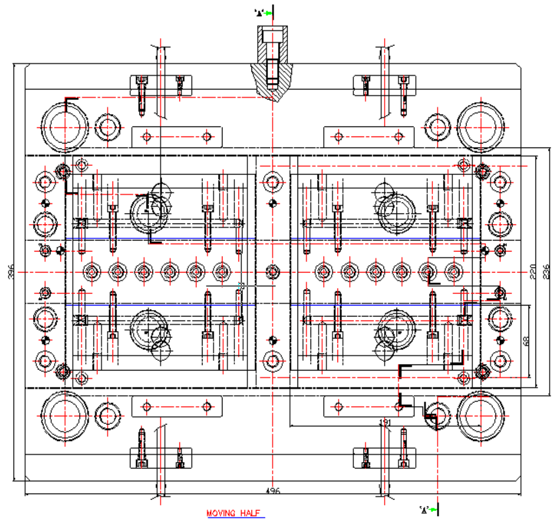


Fig.5: Top view of Assembly- Split 12 cavity Injection Mould (Moving Half)

2. Bill of Material

Bill of Material of 12-Cavity Injection Mold for Hydrogoal Dripper is shown in below. Total number of 64 parts assembled in single assembly. All parts name description, material, RM size, Hardness is given in Bill of Material Figure6.

3. Working

The Full mold in the figure below has a property of steel and includes a back plate that holds the movable side of the mold like spacer block, support plate, cavity plate and ejector mechanism to the movable platen of the injection machine. The design of the back plate for the mold has a dimension of (496mm x 220mm) and a depth of 22mm.Then it's the ejector plate often referred to as the ejector covet plate which provides backup for pins set into the ejector-retaining plate.

The retaining comes on top of the ejector plate and functions in holding ejector pins. The travel space for the ejector plates and ejector pins is provided by the spacer block. To give the plate's strength and rigidity it's smart to add a support plate. The support plate is followed by the core plate which holds the core element which is the mating half of the cavity.

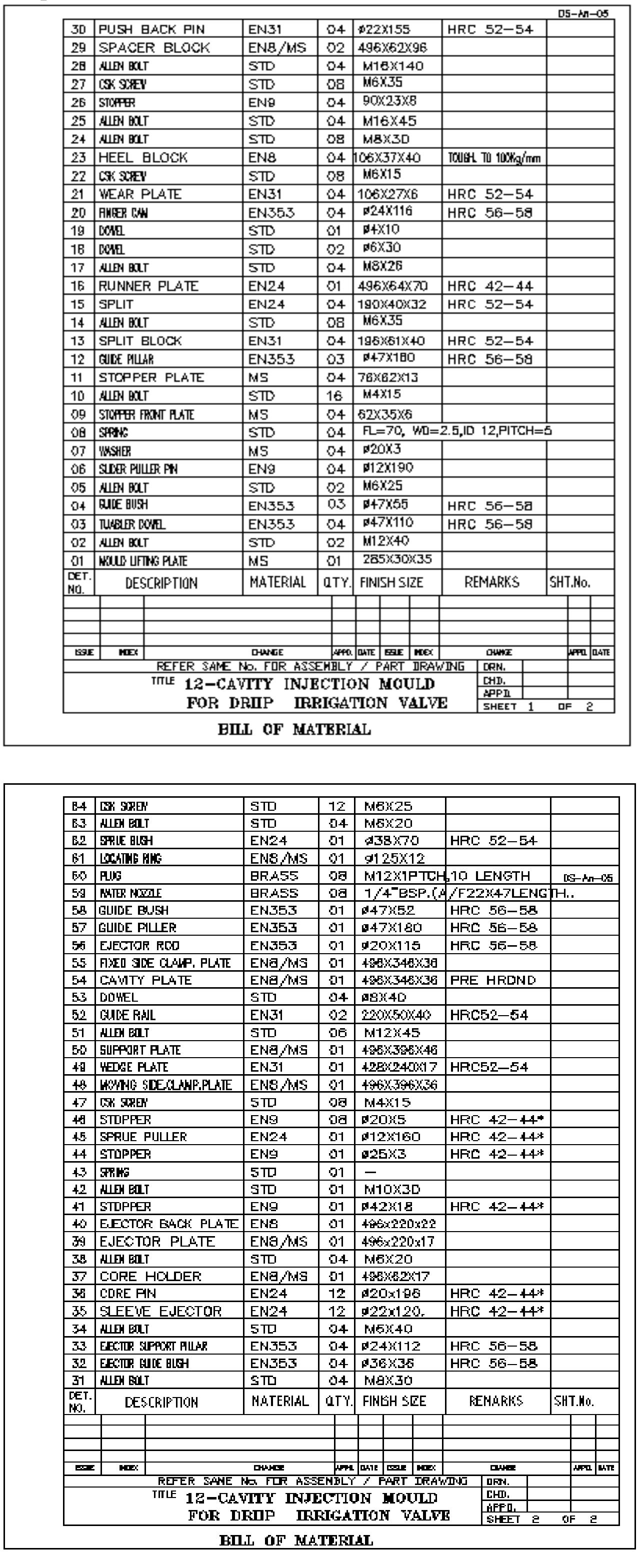
The core plate is inserted to the cavity which is where the plastic material is formed. To secure the stationary side of the mold to the machine it's necessary to have the clamping plate or also known as back plate for the core.

D. Mold flow analysis

1. Fill Time Analysis

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each color contour represents the parts of the mould which were being filled at the same time. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no color. Fill time is the time taken to fill up the part inside the cavity; it is also to show how the plastic material flows to fill the cavity. From that we know that the short shot (part of the model which did not fill) part will be displayed. Fill Time Analysis for 25% and 100% as shown in below figure no.7

Part is filling completely without any flow related problems in 0.9140s & with flow rate 43.56 Cm^3/S. Red areas highlighted in the plot indicates the EOF areas in the part.



D5-An-05

DET. NO.	DESCRIPTION	MATERIAL	QTY.	FINISH SIZE	REMARKS	SHT.No.
30	PUSH BACK PIN	EN31	04	ø22X155	HRC 52–54	
29	SPACER BLOCK	EN8/MS	02	496X62X96		
28	ALLEN BOLT	STD	04	M16X140		
27	CSK SCREW	STD	08	M6X35		
26	STOPPER	EN9	04	90X23X8		
25	ALLEN BOLT	STD	04	M16X45		
24	ALLEN BOLT	STD	08	M8X30		
23	HEEL BLOCK	EN8	04	106X37X40	TOUGH. TO 100Kg/mm	
22	CSK SCREW	STD	08	M6X15		
21	WEAR PLATE	EN31	04	106X27X6	HRC 52–54	
20	FINGER CAM	EN353	04	ø24X116	HRC 56–58	
19	DOWEL	STD	01	ø4X10		
18	DOWEL	STD	02	ø6X30		
17	ALLEN BOLT	STD	04	M8X26		
16	RUNNER PLATE	EN24	01	496X64X70	HRC 42–44	
15	SPLIT	EN24	04	190X40X32	HRC 52–54	
14	ALLEN BOLT	STD	08	M6X35		
13	SPLIT BLOCK	EN31	04	196X61X40	HRC 52–54	
12	GUIDE PILLAR	EN353	03	ø47X180	HRC 56–58	
11	STOPPER PLATE	MS	04	76X62X13		
10	ALLEN BOLT	STD	16	M4X15		
09	STOPPER FRONT PLATE	MS	04	62X35X6		
08	SPRING	STD	04	FL=70, WD=2.5,ID 12,PITCH=5		
07	WASHER	MS	04	ø20X3		
06	SLIDER PULLER PIN	EN9	04	ø12X190		
05	ALLEN BOLT	STD	02	M6X25		
04	GUIDE BUSH	EN353	03	ø47X55	HRC 56–58	
03	TUABLER DOWEL	EN353	04	ø47X110	HRC 56–58	
02	ALLEN BOLT	STD	02	M12X40		
01	MOULD LIFTING PLATE	MS	01	285X30X35		

ISSUE | INDEX | CHANGE | APPD. | DATE | ISSUE | INDEX | CHANGE | APPD. | DATE

REFER SAME No. FOR ASSEMBLY / PART DRAWING

TITLE 12–CAVITY INJECTION MOULD FOR DRIIP IRRIGATION VALVE

DRN. | CHD. | APPD. | SHEET 1 OF 2

BILL OF MATERIAL

DET. NO.	DESCRIPTION	MATERIAL	QTY.	FINISH SIZE	REMARKS	SHT.No.
64	CSK SCREW	STD	12	M6X25		
63	ALLEN BOLT	STD	04	M6X20		
62	SPRUE BUSH	EN24	01	ø38X70	HRC 52–54	
61	LOCATING RING	EN8/MS	01	ø125X12		
60	PLUG	BRASS	08	M12X1PTCH,10 LENGTH		D5-An-05
59	WATER NOZZLE	BRASS	08	1/4"BSP.(A/F22X47LENGTH..		
58	GUIDE BUSH	EN353	01	ø47X52	HRC 56–58	
57	GUIDE PILLER	EN353	01	ø47X180	HRC 56–58	
56	EJECTOR ROD	EN353	01	ø20X115	HRC 56–58	
55	FIXED SIDE CLAMP. PLATE	EN8/MS	01	496X346X36		
54	CAVITY PLATE	EN8/MS	01	496X346X36	PRE HRDND	
53	DOWEL	STD	04	ø8X40		
52	GUIDE RAIL	EN31	02	220X50X40	HRC52–54	
51	ALLEN BOLT	STD	06	M12X45		
50	SUPPORT PLATE	EN8/MS	01	496X396X46		
49	WEDGE PLATE	EN31	01	428X240X17	HRC52–54	
48	MOVING SIDE.CLAMP.PLATE	EN8/MS	01	496X396X36		
47	CSK SCREW	STD	08	M4X15		
46	STOPPER	EN9	08	ø20X5	HRC 42–44*	
45	SPRUE PULLER	EN24	01	ø12X160	HRC 42–44*	
44	STOPPER	EN9	01	ø25X3	HRC 42–44*	
43	SPRING	STD	01	–		
42	ALLEN BOLT	STD	01	M10X30		
41	STOPPER	EN9	01	ø42X18	HRC 42–44*	
40	EJECTOR BACK PLATE	EN8	01	496x220x22		
39	EJECTOR PLATE	EN8/MS	01	496x220x17		
38	ALLEN BOLT	STD	04	M6X20		
37	CORE HOLDER	EN8/MS	01	496X62X17		
36	CORE PIN	EN24	12	ø20x196	HRC 42–44*	
35	SLEEVE EJECTOR	EN24	12	ø22x120.	HRC 42–44*	
34	ALLEN BOLT	STD	04	M6X40		
33	EJECTOR SUPPORT PILLAR	EN353	04	ø24X112	HRC 56–58	
32	EJECTOR GUIDE BUSH	EN353	04	ø36X36	HRC 56–58	
31	ALLEN BOLT	STD	04	M8X30		

ISSUE | INDEX | CHANGE | APPD. | DATE | ISSUE | INDEX | CHANGE | APPD. | DATE

REFER SAME No. FOR ASSEMBLY / PART DRAWING

TITLE 12–CAVITY INJECTION MOULD FOR DRIP IRRIGATION VALVE

DRN. | CHD. | APPD. | SHEET 2 OF 2

BILL OF MATERIAL

Figure 6: Bill of Material of 12-cavity Injection Mold

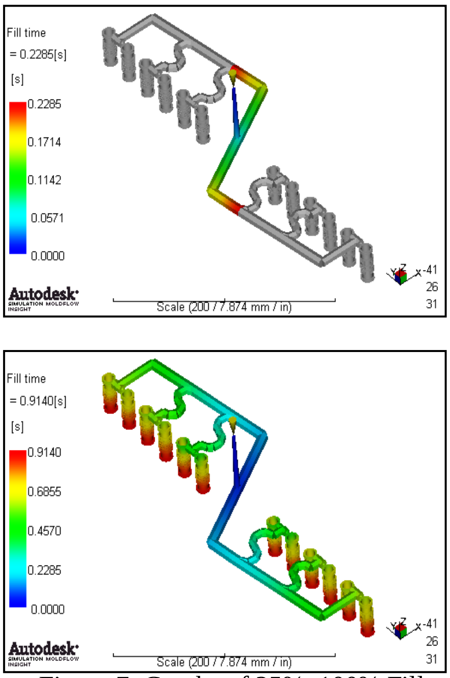


Figure 7: Graphs of 25%, 100% Fill

2. Pressure at V/P switchover

The Pressure at V/P switchover result is generated from a Fill analysis, and shows the pressure distribution through the flow path inside the mold at the switchover point from velocity to pressure control from the below plot.

Pressure Drop in the part : 26.74 Mpa

Pressure Drop in the runner system: 33.23 MPa

Total Pressure in the part & runner system: 59.97MPa

Pressure Drop is within the specified limits

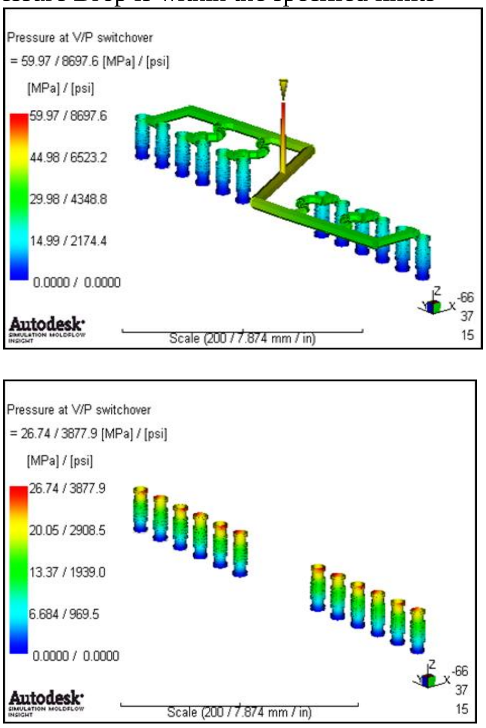


Figure 8: Pressure at V/P switchover

3. Pressure at End of Fill

The Injection pressure result, which is produced by a Fill analysis, shows the maximum injection pressure value obtained before the velocity/pressure switch-over occurs during the filling phase. The Pressure at end of fill result, which is produced by a Fill analysis, shows the pressure distribution in the cavity at the instant when the cavity is completely filled with polymer

This plot shows pressure at end of Fill. Packing pressure given is initially 47.97 MPa for 10s.

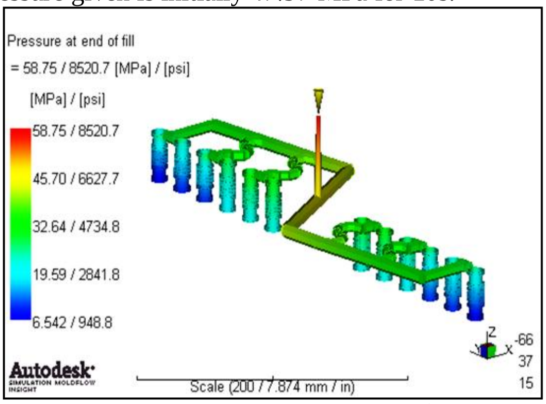


Figure 9: Pressure at the end of Fill

4. Pressure at Injection Location

The Pressure at injection location result is an XY plot generated from Fill and Microchip Encapsulation analyses.

This plot shows pressure at injection location. Packing pressure given is initially 47.97 MPa for 10s.

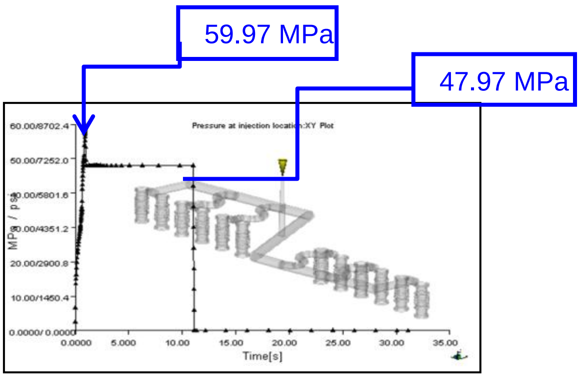


Figure 10: Pressure at Injection Location

5. Clamp Tonnage Plot

Maximum clamp tonnage required is Figure11, tonne based on the calculations done by Moldflow.

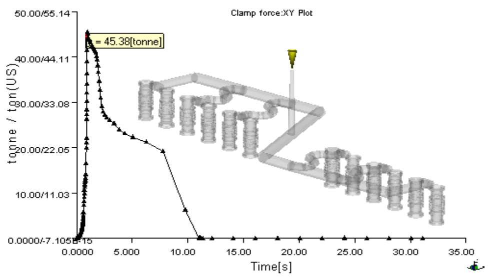


Figure 11: Clamp Tonnage Plot

6. Air Trap Analysis Result

The Air traps result is shown in figure12. shows a thin, continuous line wherever an air trap is likely to occur An air trap is where melt traps and compresses a bubble of air or gas between two or more converging flow fronts, or between the flow front and the cavity wall. Typically, the result is a small hole or a blemish on the surface of the part. In extreme cases, the air-traps should be vented out by providing air-vents in the mold

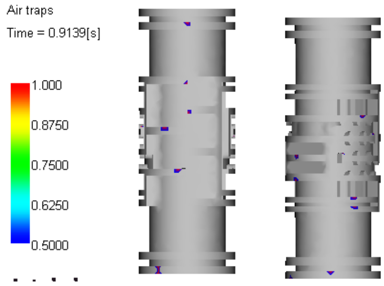


Figure 12: Air Trap Analysis

7. Volumetric Shrinkage

The average volumetric shrinkage for 3D is the average value of volumetric shrinkage over the half-gap thickness, and is plotted on the surface. This result can be used to detect sink marks on your model. High shrinkage values could indicate sink marks or voids inside the part. Volumetric shrinkage should be uniform across the whole part to reduce warpage. The plot shows the volumetric shrinkage across the component. The volumetric shrinkage ranges in between -0.9054% to 19.90%.

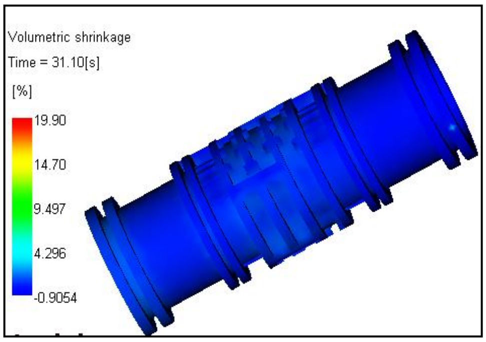


Figure 13: Volumetric Shrinkage

8. Sink Marks Analysis Result

The Sink marks is an indication of the potential shrinkage due to a hot core. It is calculated for each element at the instant when local pressure has decayed to zero during the packing stage, and reflects how much material is still melt and left unpacked. Higher Sink marks, index value shows higher potential shrinkage. However, whether or not the shrinkage would result in sink mark depends on geometry characteristics. The Sink marks, index generated indicates the likely presence and location of sink marks (and voids) in the part. This is shown in Fig. 14. In aesthetic area, shrink mark observed below 0.05mm. Sink Mark may not visible on this surface. However proper cooling provided in these areas.

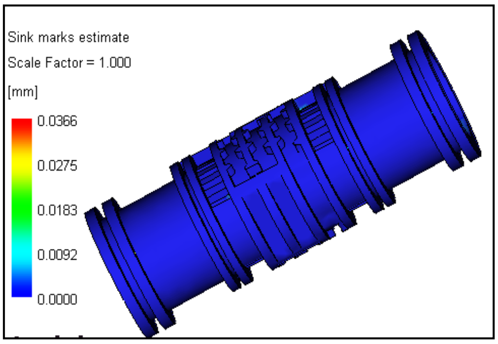


Figure 14: Sink Mark Analysis

9. Weld lines Analysis Result

The presence of weld lines may indicate a structural weakness and/or a surface blemish. The term weld line is often used to mean both weld and meld lines. The only difference between a meld line and weld line definition is the angle at which they are formed. Weld lines are formed at lower angles. Weld lines can cause structural problems, and they can also make the part visually unacceptable. Therefore weld and meld lines should be avoided if possible. However, weld lines are unavoidable when the flow front splits and comes together, around a hole, or has multiple gates. Look at the processing conditions and the weld line position to decide if the weld lines will be of a high quality. Avoid weld lines in areas which need strength, or which need to appear smooth.

The plot shows the temperature when weld line were formed. The flow front temperature drop is within the acceptable limit and hence weld lines may not be structurally strong.

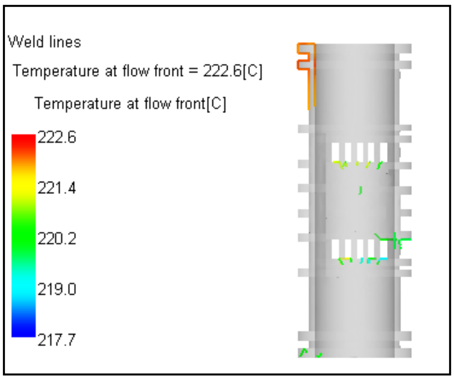


Figure 15: Weld Line Analysis

VI. Result and Discussion

On further study, discussion and elaboration of this thesis study, though the part design finally achieved is based on several trials and the underlying research theory, compatibility with melt flow analysis and mould production some amendments can be done like runner layout and gate location.

Also this customization of the part design further may result other analysis generated results of the melt flow analysis that can be fit for real world production more though what already achieved is quite feasible and practical, resulting parameters such as pressure distribution and increased flow rate. All this facts and results on the basis of actual trial, Process inspection report observed that plastic component hydrogol Dripper completely filled.

Finally, the validation of the runner design will be done by producing the component with the help of the developed mold without affecting the component's and basic mould base functionality. Flow analysis of plastic observed. Dimensional accuracy will be measured and checked with the specified dimensions. Visual and actual inspection will be done while attempting to identify the defects. Further, for fitment in the sub-assembly the component will be checked.

VII. CONCLUSIONS

In this Engineering thesis work a design and analysis of injection mould for plastic component (irrigation valve) the Software Unigraphics/Pro-E was crucial in designing all the necessary parts of the mold. In making the mold it was necessary to have the best possible product design So that it won't complicate the mold designing process. With all the necessary dimensions and by the help of 3D software product design was achieved. In this phase there were lots of ups and downs in trying to figure out what the best closing system for the irrigation valve, a lot of runner designs were drawn due pressure uniformly distribute to all 12 cavities of mold.

In this thesis it was crucial to find out if there were any defects in the manufacturing process and also finding out some important values like material selection, Fill time, Fill pattern and Clamping force.

By using the analysis software Mold flow the above values have been achieved and there were no defects found on the mould design.

VII. Future Scope

Further researches can be done on the above designs stage to put the layout of a Cooling system for the mold. The function of the cooling system of a plastic injection mold is to provide thermal regulation in the injection molding process. As the cooling phase generally accounts for about two-thirds of the total cycle time of the injection molding process, efficient cooling is very important to the productivity of the process.

References

[1] J.A. Brydson, Plastic materials, Britsh Cataloging sixth editon, 195, Buterworth Heineman Ltd.
[2] "Injection Mold Design Engineering", David Kazmer by Hanser, 2007.
[3] Bown, J., "Injection Moulding of Plastic Components", McGraw-Hil, 1979, Mould design, htp:/www. dow.com /Styron/design/guide.
[4] Technical paper by Johana Lamminmaki, Toti Lindgren, Juka Silen and Heli Vesanto, "Influence of injection moulding proces parameter on mechanical properties of polyacetal –specimen".

[5] Design Solutions Guide, htp:/www.honeywel plastics.com
[6] Technical Directory on design and Toling for plastics, CIPET, Guindy ,Chenai .
[7] Manzione,L.T., Aplications of Computer Aided Enginering in Injection Moulding,Hanser, NewYork,1987.
[8] Saman, A.M. Abdulah, A. H, Nor,MAM. Computer Simulation aportunity in Plastic injection Mould Development for Automotive part. International conference on Computer Technology and Development.
[9] Technical paper by Pankaj Shakarwal and LipinYadav' ''Design and Mould Flow Analysis of Injection Mould for Conecting Link''.
[10] Rosato, D;Donald,D;Marlene,D, 2000. *Injection molding handbook*. 3rd ed., Norwell massachusetts 02061: Kluwer Academic Publishers.
[11] UddeholmMoulding. 2004. *Steels for moulds*. [ONLINE] Available at: http://www.uddeholm.co.jp/pdf/mould-english.pdf, Sept 11.
[12] Injection mold making and molding. 2009. *Injection mold sprue and runner*. [ONLINE] Available at: http://imoldmaking.com/mold-making-2/molddesign/ runner-andgate/injection-mold-sprue-and-runner/, 21October 11.
[13] Herbert, R, *Understanding Injection Mold Design*. 1st ed. Cincinnati, USA: hanser Gardner Publications. 2001
[14] Recent methods for optimization of plastic injection molding process -- A retrospective and literature review, Author - Bharti, P. K.; Singh, Harbinder; Khan, M. I. September 2012.
[15] A 3D CAD knowledge-based assisted injection mould design system, Author: Chan, W. M.; Yan, L.; Xiang, W.; Cheok, B. T., October 2003.
[16] Moldmaker Uses an In-Depth Analytical Solution-Based Approach To Mold Flow Analysis, Author: Carrabine, Laura, August 2008.
[17] Processing parameters influencing energy efficient injection moulding of plastics and rubbers. Author: GODEC, Damir; RUJNIĆ-SOKELE, Maja; ŠERCER, Mladen. July 2012.