International Journal on Emerging Technologies **4**(1): 182-185(2013)
ISSN No. (Print): 0975-8364
ISSN No. (Online): 2249-3255

Design and Mould Flow Analysis of Injection Mould for Connecting Link

Pankaj Shakkarwal* *and LipinYadav**
**Department of Mechanical Engineering,*
Somany Institute of Technology & Management, Haryana ,India

(Received 05 May, 2013, Accepted 15 June, 2013)

ABSTRACT: Due to heavy demand in plastic products, plastic industries are growing in a faster rate. Plastic injection moulding begins with mould making and in manufacturing of intricate shape with good dimensional accuracy. To meet such requirements it is very important to adopt various advance technologies like CAD/CAM/CAE for the development of injection moulded components. A two plate injection mould design and mould flow analysis of two cavity injection mould for a given component was taken according to the customer requirement. Material selected for "CONNECTING LINK" was Poly Acetal. The 3-D model of the component and extraction of core and cavities was performed in Pro-engineer (Mould wizard) software. Auto Desk Mould Flow Analysis software is a powerful simulation tool to locate gate location and predict the defects in the component.

Keywords: Injection Moulding, Tool Design, MoldFlow Analysis, Moulding Defects

I. INTRODUCTION

Now a day, the technology of the Tool and Die manufacturing is one of the fastest growing industries in the world. Plastic is the most used material in the world. Among various plastic production technologies, Injection Moulding takes up approximately 32%. In injection moulding, the Tool Design of Mould is of Critical importance for the quality and economy. Injection moulding is a process that forms the plastic into desired shape by melting the plastic material and forcing the plastic material under pressure into the mould cavity. Now a day there is a need for improving the process to increase productivity, reduce the cycle time in injection moulding. In order to achieve the processing parameters, they commonly follows on experience, hit and trail method due to which this process is not practical for complex models. As new generation designers require more powerful software to analyse and to optimize injection moulding process by manipulating parameters to reduce cycle time. However development of CAD/CAM/CAE technology especially Mould flow Analysis, the number of trails on mould can be reduce to achieve good quality product. In this paper, Mould Design for CONNECTING LINK which is a part of latch assembly used in automobile door's has been design in high end CAD software Pro-Engineer, Mould flow analysis was performed on the component for finding out moulding defects and reducing development time and expense.

A. Design Methodology

This paper presents a practical Tool Design procedure/methodology of an injection mould.

(i) Component Modeling: As per customer requirement to produce CONNECTING LINK plastic component by injection moulding process. The mould for producing the above mentioned component was modelled by using part design module of Pro-Engineer software.

(ii) CAD for Mould Design: Before starting mould design the designer should be in possession of the following information's.

- A fully detailed component drawing.
- Specification of the moulding material including grade and colour.
- Moulding machine specification.
- Number of impression.
- Shrinkage of the material.
- Type of mould e.g Two or Three plate
- Type of runner or gate
- Parting line of mould
- Types of ejection system.
- Type of cooling system.
- Injection pressure.
- Shot weight.
- Distance between the tie bars.
- Shut height of mould.
- Shut height of machine.
- Clamping force.

The assembly of injection mould design is proposed on practical design parameters. CAD/CAM helps designers to speed up design process and reduce the long lead time. The critical parameters to be considered while designing in injection mould tool are:

- Shrinkage
- Draft angle

- Number of cavities
- Selection of parting surface
- Feed system
- Ejector system
- Venting
- Cooling system

The 3D model and Extraction of Core and cavity is performed using pro-engineer and 2D detailed drawing are drafted in Auto Cad for the manufacturing the tool in shop floor.

(iii) Cost Estimation: The main elements of cost are as follows:

- Raw Material Cost
- Machining Cost
- Heat Treatment Cost
- Cost of Standard Items
- Design Cost
- Assembly & Try-out Cost
- Administrative Overheads

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

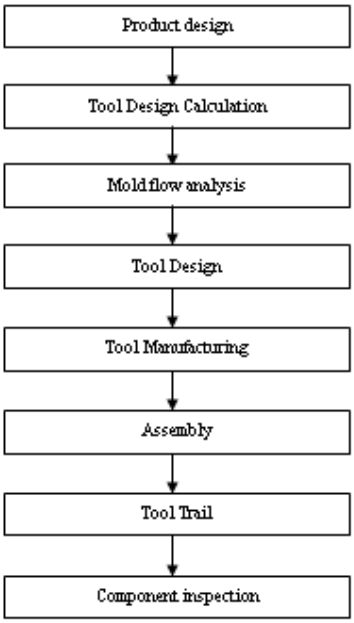

Fig.1. Flowchart for practical Tool Design and Manufacturer.

II. LITERATURE REVIEW

Plastic is one of the most versatile materials in the modern age which is widely used in many products in different shapes which are moulded through the application of heat and pressure [1]. Injection moulding has become the most important process for manufacturing plastic parts due to its ability to produce complex shapes with good dimensional accuracy [2]. The injection moulding process involves feeding raw material, plasticize the raw material, fill the mould, pack the mould, hold pressure, cooling of mould and lastly opening of mould and Part ejection [3].The main factor in the injection moulding are the temperature and pressure history during the process, the orientation of flowing material and the shrinkage of the material. The raw material is generally fed through an angular type sprue channel which feeds the resin pellets forward inside the heated barrel [4]. To start-up a new mould design, the designer should know some important points to avoid some mistakes before going further. i.e., Product outlook design, material usage, correction shrinkage of the material, number of cavities and selection of mould base. In injection moulding, there is an optimum gate size and it should large enough for suitable fill rate and small enough seal off and prevent back flow or over packing [5].

Cad/ Cam can help designer to speed up design for the plastic part and mould design process and reduce the long lead time [6].The introduction of simulation software has made a significant impact in the injection moulding industry. With the increasing use of computers in design engineering, the amount of commercially available software on the market has also increased [7].

Traditional trial runs on the factory floor can be replaced by less costly computer simulations. Now a day's, research on optimizing the plastic injection moulding process has developed a lot.

CAD/CAE tools are used to produce an optimal mould gating design using Catia and Mould flow applications. The mould flow analysis helps in reducing costs and time and also prevents other defects occurring in the process [8].

III. PROBLEM FORMULATION

In present scenario, the most of the plastics industries are under great pressure, due to the globalization of the market, and high demand of product. In injection moulding, the design of mould is of critical importance for the product quality and efficient processing, which is also responsible for the economics of the entire process. Mould designers are required to possess thorough and broad experience, because the detail decisions require the knowledge of the interaction of various parameters. From the last five decades, SME's uses hit and trail method for designing and analysis purpose that cause wastage of large amount of plastics material to get the desired parameters to setup the component and machine.

IV. ANALYSIS

Analysis is essential for designing and mould making through simulation step-up and result interpretation to show how changes to wall thickness, gate location, material and geometry affects manufacturability and also experiments with "what-if" scenarios before finalizing a design.

A. Mould Flow Analysis

The Mould flow analysis was performed using Autodesk Mould Flow analysis software. The sequence of work involved in Mould flow analysis is given under.

1. Converting the 3D model in STEP OR IGES format.
2. Meshing the model by using dual domain type of mesh.
3. Importing the meshed file to the solver package specifying the boundary condition, loads such as injection pressure, injection time, mould temperature, melt temperature, material properties etc.
4. Building the feed system such as sprue, runner and gate.
6. Mesh the feed system and cooling lines.
7. Run the analysis for different analysis types like fill, flow, warpage etc.
8. Study the result, interpret them.
9. Establish the optimized data for runner, gate, sprue dimensions, coolant temperature etc. Based on the analysis the optimal combination of part geometry, material choice, gate location and process parameter to produce quality finish part are determined. This analysis also gives the result of fill analysis, pack analysis, warp analysis and cooling analysis.

V. MOULD FLOW ANALYSIS RESULT

A. Gate Location Analysis

Placing a gate appropriately is the most critical factor in determining the quality of the part. The analysis result, the gate location on the part may be preset or appeared with two or three choice; then the optimum gate locations may need to be examining by running the filling analysis on different best gate locations. Figure 2 shows the result of gate location. Blue area represents the best gate locations for the part.

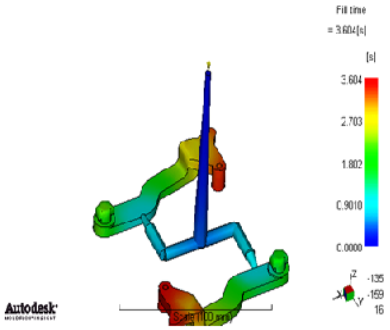

Fig. 2. Gate Location.

B. Fill Time Analysis

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each colour 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 colour. 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..

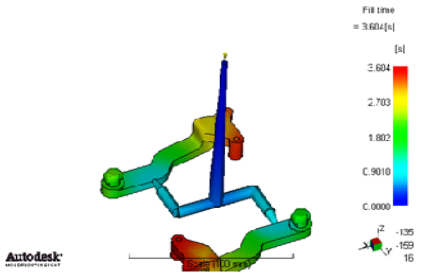

Fig. 3. Fill Time.

C. Air Trap Analysis Result

The Air traps result is shown in figure4. 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 compression increases the temperature to a level that causes the plastic to degrade or burn. Move the injection locations so that the air traps form in easy-to-vent areas, such as at the parting plane.

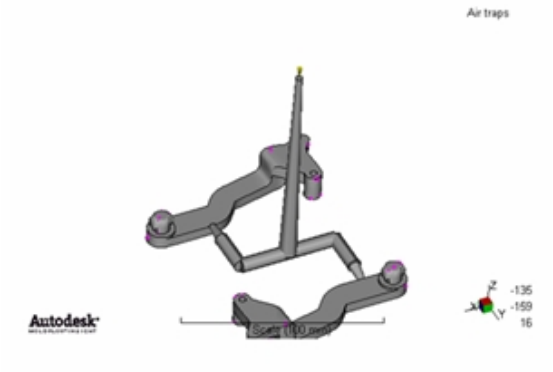

Fig. 4. Air Trap.

D. Weld lines Analysis Result

The Weld Lines result in Figure5. displays the angle of convergence as two flow fronts meet. 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.

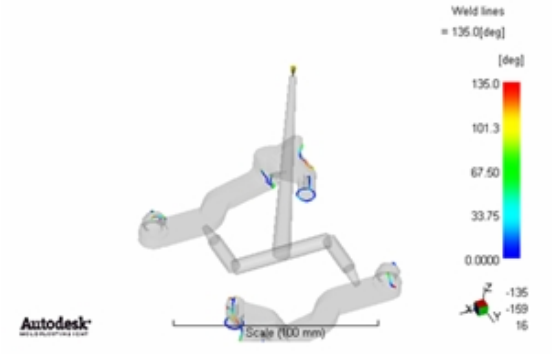

Fig. 5. Fill Time.

E. Sink Marks and Index 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.

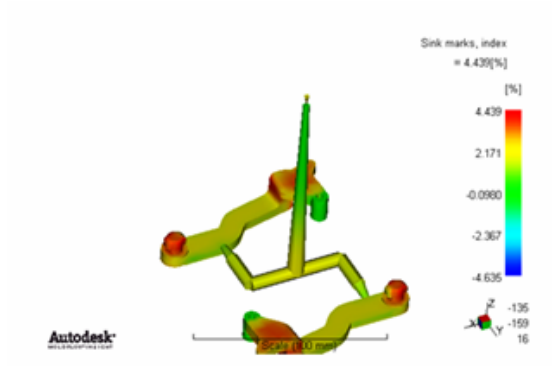

Fig. 5. Sink Marks.

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. 5.

CONCLUSION

The present study reveals that the Design with CAD Tools and Mould Flow Analysis is the important requirements for injection moulding components. The study results in reduction of wastages and saved valuable man hours, that will be a great loss in terms of money occurs during production phase in SME's before. The analysis work shows the parameters such as sink marks, fill time, weld line, air traps etc. that will affects the quality of the finished product. The analysis also results in minimum moulding defects in injection moulded components. Thus, it was concluded that the mould flow software is a preventive and corrective tool, helps the engineer to analyse the process to decrease the cycle time and to improve the Quality of the Product.

REFERENCES:

[1]. J.A. Brydson, Plastic materials, British Cataloging sixth edition, 1995, Butterworth Heinemann Ltd.

[2]. Bown, J., "Injection Moulding of Plastic Components", *McGraw-Hill*, 1979.

[3].Mould design,http://www.dow.com/Styron/design/guide.

[4]. Technical paper by Johanna Lamminmaki, Totti Lindgren, Jukka Silen and Heli Vesanto, "Influence of injection moulding process parameter on mechanical properties of polyacetal –specimen".

[5]. Design Solutions Guide, http://www.honeywell plastics.com

[6]. Technical Directory on design and Tooling for plastics, CIPET, Guindy , Chennai .

[7]. Manzione,L.T., Applications of Computer Aided Engineering in Injection Moulding,Hanser, NewYork,1987.

[8]. Saman, A.M. Abdullah, A.H,Nor,MAM. Computer Simulation apportunity in Plastic injection Mould Development for Automotive part. *International conference on Computer Technology and Development.*