OPTIMUM GATE LOCATION IN PLASTIC INJECTION MOULDING USING COMPUTER SIMULATION

MADIGUNDU SREEKANTH *RAGHUVEER*[1] *(Department of Mechanical Engineering, G.P.R College of Engineering, Andhra Pradesh, India)* Mobile:8374800885,Email:raghu.mech.338@gmail.com

P.HARI SANKAR[2], associate professor *(Department Of Mechanical Engineering, G.P.R College of Engineering, Andhra Pradesh, India)*

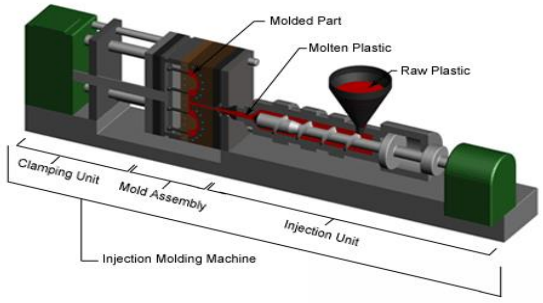
Mobile:9885185687, Email: harisankar_pallah@rediffmail.com

Abstract: Due to heavy demand in plastic products, plastic industries are growing in a fastest rate. Plastic injection moulding begins with mould making and in manufacturing of complex shapes. Optimal setting up of injection molding process variables plays a very important role in controlling the quality of the injection molded products. It is all the most important to control attribute defects like sink marks. Sink marks are basically a "designed in" problem and hence it is to be attended during designs stages. Owing to certain conditions and constraints, sometimes, it is rather ignored during design stages and it is expected to be handled by molders with only instruction to 'do the best'. Handling of numerous processing variables to control defects is a mammoth task that costs time, effort and money. Mould Flow analysis is a powerful simulation tool to minimize the sink marks and to predict the production time required at the lowest possible cost. Verification using simulation requires much less time to achieve a quality result, and with no material costs, as compared with the conventional trial-and-error methods on the production floor.

In this thesis, a comparative analysis has been performed by taking different process parameters and single gate, two gate and three gate locations to minimize the sink marks in the manufacturing of EMF Load Cell used in weighing machine. Modeling is done in Pro/Engineer and Mould Flow Analysis is done in Plastic Advisor in Pro/Engineer.

I. INTRODUCTION TO INJECTION MOULDING:

Injection molding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide variety of products are manufactured using injection molding, which vary greatly in their size, complexity, and application. The injection molding process requires the use of an injection molding machine, raw plastic material, and a mold. The plastic is melted in the injection molding machine and then injected into the mold, where it cools and solidifies into the final part.

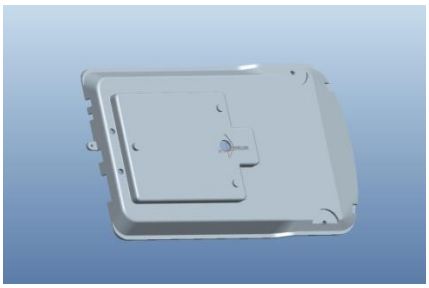


Injection molding is used to produce thin-walled plastic parts for a wide variety of applications, one of the most common being plastic housings. Plastic housing is a thin-walled enclosure, often requiring many ribs and bosses on the interior. These housings are used in a variety of products including household appliances, consumer electronics, power tools, and as automotive dashboards. Other common thin-walled products include different types of open containers, such as buckets. Injection molding is also used to produce several everyday items such as toothbrushes or small plastic toys. Many medical devices, including valves and syringes, are manufactured using injection molding as well.

II. MOULD FLOW ANALYSIS:

Mould flow, 3D solids-based plastics flow simulation that allows plastics part designers to determine the manufacturability of their parts during the preliminary

design stages and avoid potential downstream problems, which can lead to delays and cost overruns. Following are the benefits: In the simulation the EMF load cell part is used .



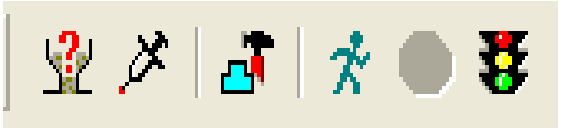
- Optimize the part wall thickness to achieve uniform filling patterns, minimum cycle time and lowest part cost Identify and eliminate cosmetic issues such as sink marks, weld lines and air traps.
- Determine the best injection locations for a given part design

Mould flow analysis gives you the ability to maintain the integrity of your product designs. It provides you the tools to quickly optimize part designs and check the impact of critical design decisions on the manufacturability and quality of the product early in the design process.

III.SOFTWEAR USED IN SIMULATION:

PLASTC ADVISOR: Plastic advisor is an add on analysis package for Pro/Engineer, especially for plastic injection moulding. For doing this analysis, after drawing the required object Select applications > plastic advisor > pick datum point for injection location > ok.

There will be an analysis tool bar



The icons available are

1) moulding parameter icon
2) specify injection location
3) special analysis icon
4) Start analysis icon.

Table 1. the below table shows the input parameters used for analysis.

	CASE 1	CASE 2	CASE 3
INJECTION PRESSURE (MPa)	180	200	230
MOLD TEMPERATURE (^{0}C)	40	60	80
MELT TEMPERATURE (^{0}C)	200	230	300

IV.ANALYSIS RESULTS:

Results obtained are

1) Plastic flow
2) Fill time
3) Confidence of fill
4) Injection pressure
5) Pressure drop
6) Flow front temperature
7) Quality prediction
8) Weld lines and air traps.

- Fill time result shows the flow path of the plastic through the part by plotting contours which join regions filling at the

same time. These contours are displayed in a range of colors red indicates the first region to fill, blue to indicate the last region to fill. A short shot is a part of the model that did not fill, will be displayed as translucent.

- This result shows how the injection pressure distribution throughout the mold at specific time.

- Pressure drop is opposite to that of the injection pressure. This result shows how the pressure is dropped throughout the cavity at the specific time.
- The confidence of fill result displays the probability of a region within the cavity filling with plastic at conventional injection molding conditions. This result is derived from the pressure and temperature results.
- The flow front temperatures represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.
- Weld lines result indicates the presence and location of weld and weld lines in the filled part model. These are places where two flow fronts have converged.
- The air trap result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped. The regions highlighted in the result are positions of possible air traps.

- Confidences of fill results indicate that there is a low Confidence of Fill due to the high pressure required to fill the cavity. The advice topic on pressure drop indicates that to fill the part you will need to change one of the following things:

    1) material
    2) injection location
    3) injection pressure
    4) part geometry
    5) injection pressure

- The Quality Prediction plot uses several results to give an indication of the overall quality of the part. When the Quality Prediction result indicates a problem the following things have to be checked and changed if necessary.

    1) Shear stress is too high
    2) Cooling time is too high
    3) Pressure drop is too high
    4) Flow front temperature is too high
    5) Flow front temperature is too low

- Weld lines are not all critical and do not always lead to visual or structural defects. Weld lines are often unavoidable but we can change the flow of the material to place them in areas that are less sensitive. Weld lines should only be in areas that are not visually or structurally sensitive

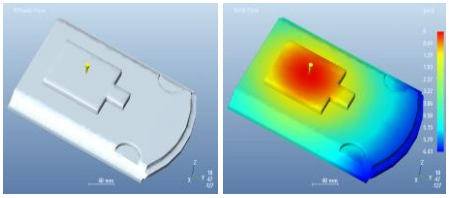
- Air traps appear when flow fronts meet, trapping pockets of air. Like weld lines these are also unavoidable so we should take care that air traps should only be in locations where the air can be vented

- The Confidence of Fill and the Quality Prediction results are good starting points when checking an analysis. They will show you where there may be problems with the part.

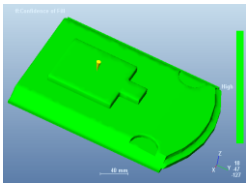
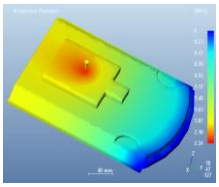
- If flow front is too low in an area where weld lines are present, the weld lines may appear worse. In areas where the flow front temperature is too high, material degradation and surface defects may occur. We must make sure that the flow front temperature is always within the recommended temperature range for the material used.

GATE :1,

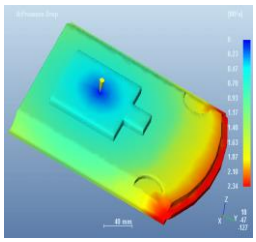
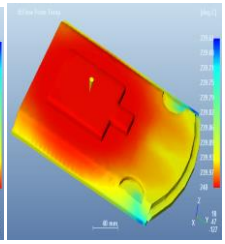
CASE :1



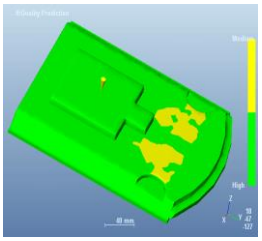
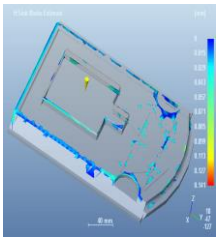
Plastic flow Fill time

Confidence of fill injection pressure

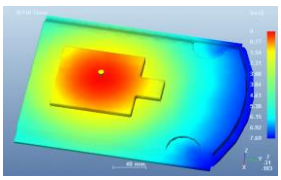



Pressure drop flow front temp

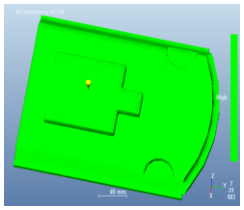
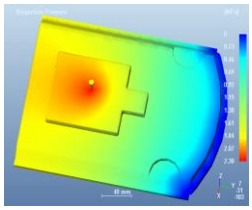



Quality prediction sink marks

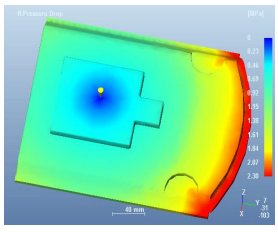
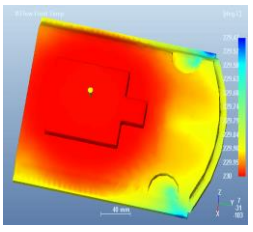
CASE:2

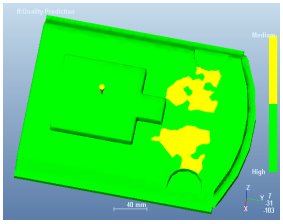
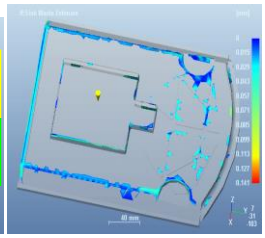
Plastic flow fill time

Confidence of fill injection pressure

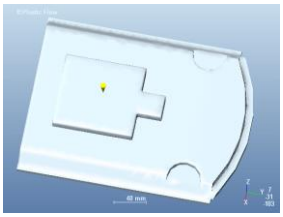
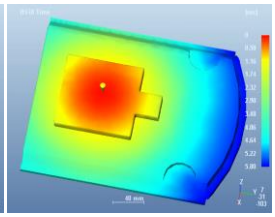



Pressure drop flow front temp

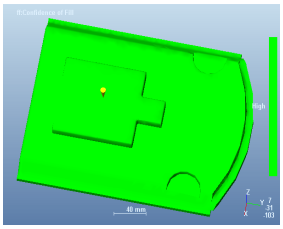
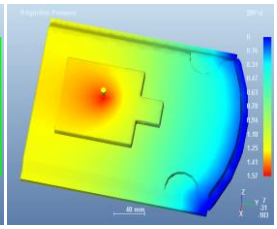



Quality prediction sink marks

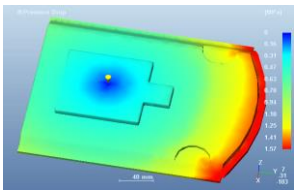
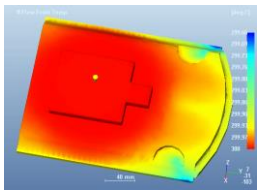
CASE :3

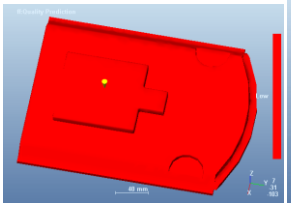
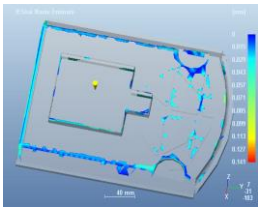
Plastic flow fill time

Confidence of fill injection pressure

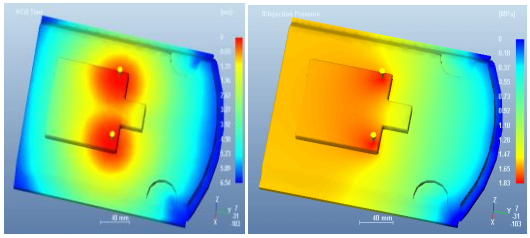
Pressure drop flow front temp

Quality prediction sink marks

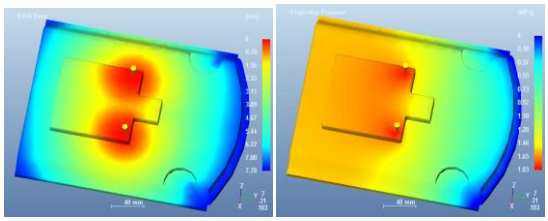
GATE:2

CASE :1



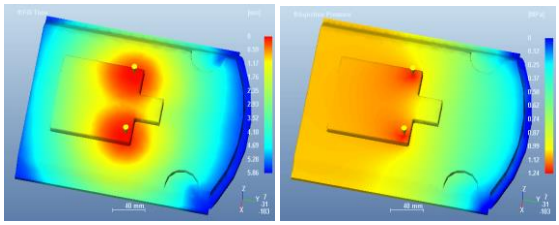
fill time injection pressure

CASE:2



Fill time injection pressure

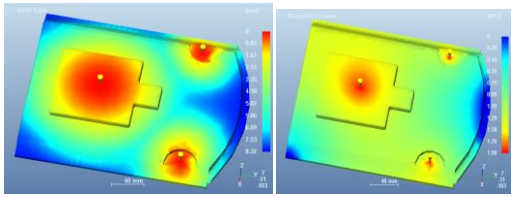
CASE:3



Fill time injection pressure

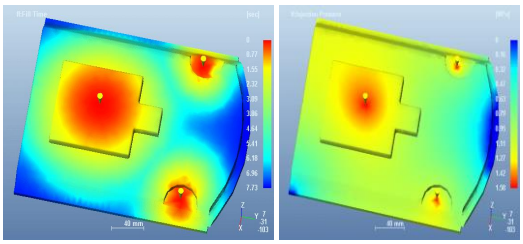
GATE-3

CASE-1



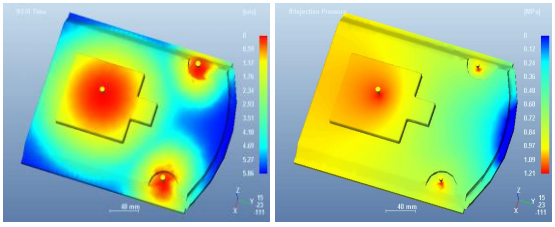
Fill time injection pressure

CASE:2



Fill time injection pressure

CASE:3



Fill time ***injection pressure***

V. RESULTS TABLE:

GATE:1

	Case 1	**Case 2**	**Case 3**
Confidence	medium	medium	medium
Fill time (secs)	6.43	7.69	5.80
Injection pressure (mpa)	2.34	2.30	1.57
Weld lines	yes	no	no
Air traps	yes	yes	yes
Cycle time (secs)	126.59	147.85	212.22
Quality prediction	medium	medium	low
Sink marks	8 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.

GATE:2

	Case 1	Case 2	Case 3
Confidence	medium	medium	medium
Fill time (secs)	6.54	7.78	5.86
Injection pressure (mpa)	1.83	1.83	1.24
Weld lines	yes	yes	yes
Air traps	yes	yes	yes
Cycle time (secs)	126.19	147.41	212.02
Quality prediction	medium	medium	low
Sink marks	8 % of your model was found to be prone to sink marks.	8 % of your model was found to be prone to sink marks.	8 % of your model was found to be prone to sink marks.

GATE:3

	CASE 1	CASE 2	CASE 3
Confidence	medium	medium	medium
Fill time (secs)	8.37	7.73	5.86
Injection pressure (mpa)	1.97	1.58	1.21
Weld lines	yes	yes	yes
Air traps	yes	yes	yes
Cycle time (secs)	111.92	147.30	212.01
Quality prediction	low	medium	low
Sink marks	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.

VI. CONCLUSION:

In this thesis, the optimal process parameters and the optimal number of gates required to fill the component EMF load cell with least defects is analyzed. The number of gates taken is one, two and three. The process parameters considered in three cases, **Case-1:** Max Injection Pressure: 180MPa, Mold Temperature: 40 deg C, Melt Temperature: 200 deg C, **Case-2:** Max Injection Pressure: 200MPa, Mold Temperature: 60 deg C, Melt Temperature: 230 deg C and **Case-3:** Max Injection Pressure: 230MPa, Mold Temperature: 80 deg C, Melt Temperature: 300 deg C.

The material is PP(poly propylene).

By observing the analysis results of PP, use of two gates is better since 8 % of model was found to be prone to sink marks when two gates is used but when single gate or three gates are used 9% of model was found to be prone to sink marks respectively. When the process parameters are considered, considering Case 1 and Case 2 parameters is better the quality prediction is medium.

VII.REFERENCES:

1. Plastic Injection Molding: Manufacturing Process Fundamentals by Douglas M. Bryce
2. Injection Mould Design: A Design Manual for the Thermoplastics Industry by R G W Pye
3. The Mould Design Guide by Peter Jones
4. Injection molding - Society of Plastics Engineers
5. Injection moulding of plastic components by John Bown
6. Rosato, D. Injection Moulding Handbook. Second Edition, Chapman & Hall, London, 1995.
7. Lam, Y.C., Jin, S. Optimization of the gate location for plastic injection moulding. Journal of Injection Moulding Technology, 2001
8. Sahputra, I.H. Comparison of two flow analysis software for injection moulding tool design. Proceedings of the International Conference on Industrial Engineering and Engineering Management, 2-4 Dec. 2007, Singapore.
9. Saman, A.M., Abdullah, A.H., Nor, M.A.M. Computer simulation opportunity in plastic injection mould development for automotive part. International Conference on ComputerTechnology and Development, 13-15 Nov. 2009, Kota Kinabalu, Malaysia
10. Journal Paper of Application of computer simulation for finding optimum gate location in plastic injection moulding process by S.R. Pattnaik, D.B. Karunakar, P.K. Jha.

