Journal of Engineering Sciences
Vol 13, Issue 03, MARCH/2022
ISSN NO:0377-9254

MINIMIZATION OF SINK MARK DEFECTS IN INJECTION MOLDING PROCESS OPTIMIZATION OF A COMPOSITE DRIVE SHAFT USED IN AUTOMOBILES

MATAM KUMARA SWAMY[1], V.VENKATA SIVA REDDY[2], N. PHANI RAJA RAO[3]
[1]PG student, Dept. of Mechanical Engineering, SVIT, Anantapur.
[2]Assistant Professor, Dept. of Mechanical Engineering, SVIT, Anantapur.
[3]Assistant Professor & Head of the department, Dept. of Mechanical Engineering, SVIT, Anantapur.

ABSTRACT

The purpose of this paper is to present an integrated approach for improving the quality characteristics of the injection molded part (T-block/Thin Structured Part) being manufactured at manufacturing industry, where the rejection rate for T-block/Thin Structured Part was on ramp due to sink marks defects. The proposed integrated approach embraces the concept of Design of Experiments and Response surface design methodology for injection molding process optimization. The Taguchi Method(TM) was used to short list the variables that have significant effects on the sink marks in injection molded parts. Furthermore, the optimization approach of Response Surface Methodology (RSM) was utilized for the experimental research to acquire a prediction model that can be used to optimize injection molding process in terms of fine sink marks reduction. The result shows that the sink marks reduction predicted by the integration of the Taguchi Method and RSM indeed decreased from 0.0088 (Taguchi's result) down to 0.0080 mm. The empirical results reveal that the integration of the Taguchi Method and RSM could effectively improve the quality.

Keywords: Sink mark, plastic injection molding, Taguchi optimization, process optimization, attribute defects in injection molding.

1. INTRODUCTION

Injection molding is the most significant process to produce plastic products. It produces most intricate complex geometry parts effortlessly. The parts which are manufactured by the injection molding contain some defects and these defects influences the quality of the component. Chao-chyun An, Ren-Haw Chen [1], in their paper have suggested that, defect of flow marks which can be easily controlled when mold temperature and injection speed are comparatively low. Lei Xie, Grehard Ziegmann [2], have studied comprehensively the mechanical properties of weld line defect for polypropylene (PP) composites. Mehdi Moayyedin et al.[3], presented an analysis of short shot possibility in injection molding process. They have adopted the Taguchi method to identify the significant process and geometric parameter to overcome short shot occurrence. Daniele Annicchiarico et al.[4] in their paper have developed a methodology to overcome shrinkage defects. Their analysis results showed that the methodology adopted was capable of detecting the factors which will have significant effect on shrinkage at a micro scale level. ShinChin Nian et al.[5] in their paper have described that the inconsistent thickness in component geometry, poor spruce, runner, gate sand improper molding condition settings may cause plastic parts to warp excessively. Xianjun Sun et al.[6] have opinioned that it is a challenge to produce parts that do not warp and have developed technology to measure part war page quickly and accurately with digital image correlation method.

They have told that the mold temperature has a significant effect on warpage. Yu wang et al.[7] in their study, have examined the various cooling techniques in injection molding process. The spiral and conformal technique, cooling design of channels are able to achieve uniform mold cooling so that molded parts shape is maintained. MingShyan Huang et al.[8] have examined the influence of clamping force on tie bar elongation and have opined that a small clamping force may produce defects such as flash and poor geometric accuracy. The large force results in insufficient air venting during mold filling, leading to the generation of the short shot. Ming-Shyan Huang and ChengYou Lin [9], in his paper he has developed a novel searching algorithm based on information about tie-bar elongation with various clamping force settings to identify the proper clamping force value to set. Xundao Zhou et al.[10] have studied feature extraction and physical

Vol 13, Issue 03, MARCH/2022
ISSN NO:0377-9254

interpretation of melt pressure during the injection molding process and concluded that melt pressure is very crucial in injection molding. They installed variety of pressure sensors in injection molding machines and molds to collect melt pressure information and further inferred that challenges exist for these feature extraction.

2. MATERIALS

The paper is involved in optimization of IM process through multiple potentially correlated control variables, by searching the entire search space for potential and feasible combination of process control parameter values for minimum value of sink marks. The design integrated method predicts good response in terms of dimensional part's accuracy requirements (minimum sink marks), and can improve the Honda upper part mold ability. The method is feasible and has potential to be easily adopted in industrial product/process development to define the optimal process parameters. The below Figure gives an over view of above approach.

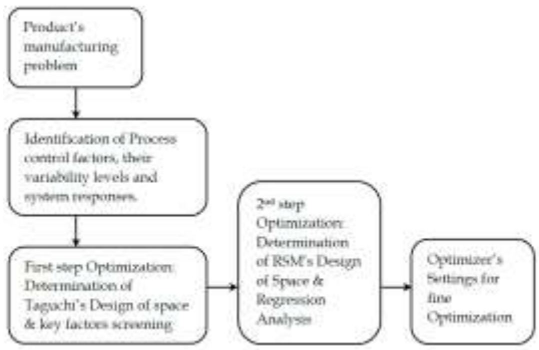


Figure: Flow Diagram for Optimization of IM Process Response

The first step towards optimization focuses on factors screening based on Taguchi experimental design, to evaluate main effects and interactions. The main purpose of a DOE is to determine level of selected factors that leads to optimum performance. The selected factors are data inputs for the next step. The second optimization step is based on RSM plan to sample the space design. The data were analyzed, and the fitting model was applied to study the interactions among factors. Based on above an optimal set of values was determined for each factor by tuning the parameters in their variability range. The output is a linear cum regression model that describes the system responses as functions of the selected factors. The developed regression model describe the process responses as functions of key factors and an objective optimization is proposed by tuning the process factors in their variability range.

Injection molding (IM) consists of high pressure injection of the raw material into a mold which shapes the polymer into the desired shape. In this process typically palletized thermoplastic raw material is fed through a hopper into a heated barrel with a reciprocating screw. On entrance to the barrel the thermal energy increases and the Vander Waals forces that resist relative flow of individual chains are weakened as a result of increased space between molecules at higher energy states. Due to this viscosity reduces and it enables the raw material to flow with the driving force of the injection unit. The reciprocating screw enables the mechanical shearing of the material and add significant amount of frictional heating to the raw material. The injection molding process can be divided into three phases. Filling phase, the most important phase as plastics flow into cavity. Pressurization phase: it begins after the cavity has just filled. However the edge and corners of the cavity may not contain plastic. Compensating phase: plastics have a high volumetric shrinkage, around 25% from average melt temperature to solid. It requires more material must be injected into the cavity to compensate for the plastic shrinkage as it cools.

Table - 1: Summary – Mesh (T – shape)

ITEM NAME	ITEM DATA
Mesh type	Normal
Part dimension	80.00x85.00x80.00 mm
Mold dimension	80.00x95.00x80.00 mm
Cavity (Part) volume	280000000.000 mm^3
Number of elements	268348
Number of part elements	268348
Number of nodes	283539
Polymer type	**ABS**
Grade name	POLYFLAM RABS 90000 UV5
Producer	A. Schulman
Melt flow index	Unavailable
Fiber percent	0.00 %
Melt temperature range	220 ~ 250 °C
Mold temperature range	40 ~ 60 °C

Ejection temperature	100 °C
Freeze temperature	120 °C
Item Name	**Item Data**
Filling time	2.630 sec
Melt temperature	235.000 °C
Mold temperature	50.000 °C
Maximum machine pressure	155.000 Mpa
Packing time	17.100 sec
VP switch by filled volume (%)	98.000 %

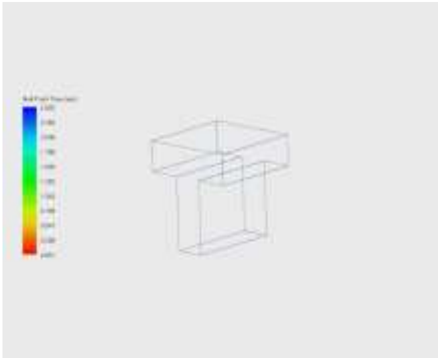
Table - 2: Summary – Mesh (T – shape top side)

Item Name	Item Data
Mesh type	Fine
Part dimension	100.00x50.00x80.00 mm
Mold dimension	100.00x60.00x80.00 mm
Cavity (Part) volume	144000000.000 mm^3
Number of elements	502750
Number of part elements	502750
Number of nodes	539891
Polymer type	**ABS**
Grade name	POLYFLAM RABS 90000 UV5
Producer	A. Schulman
Melt flow index	Unavailable
Fiber percent	0.00 %
Melt temperature range	220 ~ 250 °C
Mold temperature range	40 ~ 60 °C
Ejection temperature	100 °C
Freeze temperature	120 °C
Item Name	**Item Data**
Filling time	2.390 sec
Melt temperature	235.000 °C
Mold temperature	50.000 °C
Maximum machine pressure	155.000 Mpa
Packing time	15.400 sec
VP switch by filled volume (%)	98.000 %

3. RESULTS AND ANALYSIS

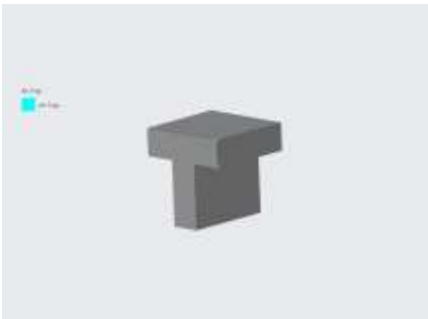
Melt front advancement is a position indicator as melt front boundary movement in different time duration in the filling process. From the melt front advancement one can:

- Examine imbalances in the filling pattern of the molding.
- Check for incomplete filling of cavity or short shot problem.
- Identify the weld line locations.
- Identify the air trap locations.
- Check the flow contribution of each gate for a balanced runner.
- Check if the gate location is correct to balance the flow and eliminate weld line.



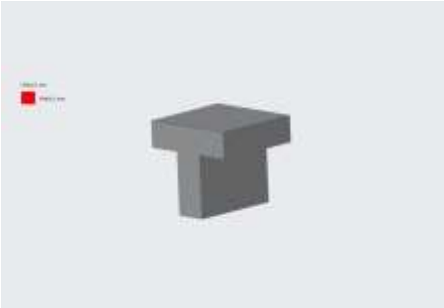
The Yellow or Red regions show the areas where the melt filled with difficulty, which could result in quality related problems.

Air Trap result shows the possible locations where air trap can occur.



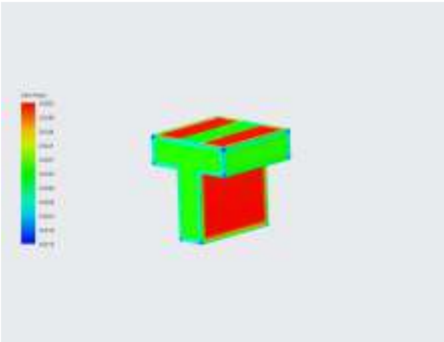
Vol 13, Issue 03, MARCH/2022
ISSN NO:0377-9254

Weld Line result shows the weld lines that indicate potential spots of weaker structure. The darker the weld line, the weaker the structure.



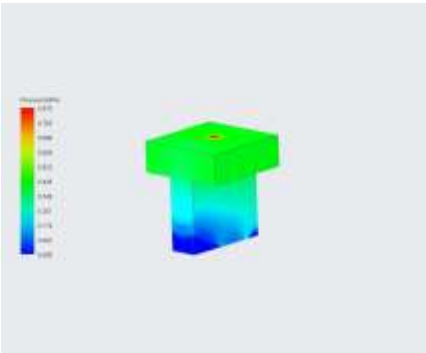
Sink Mark is an index to evaluate the packing effect.

- Positive value—shows insufficient packing. This may cause depression or sink marks on the surface of the molded part.
- Negative value—shows over packing
- Value close to zero—shows correct packing

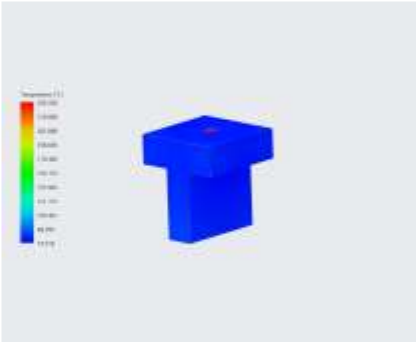


Pressure result shows the pressure distribution of the plastic at the end of filling. Use the pressure result to analyze the following:

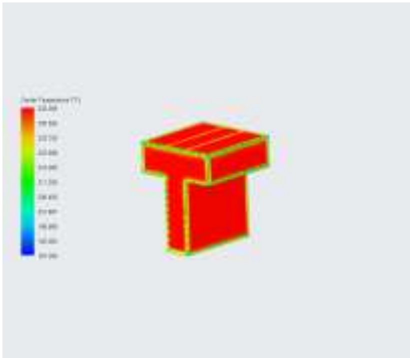
- Check the pressure transmission conditions.
- Check for drop in pressure in the runner system.
- Check the mold design for a balanced flow.
- Check for over packing and flashing of the melt.
- Examine the extent of packing or holding.



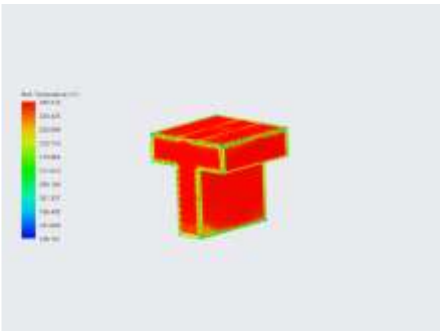
Temperature result shows the plastic melt temperature distribution at the end of filling. The temperature distribution shows temperatures in all three dimensions for the fully cavity.



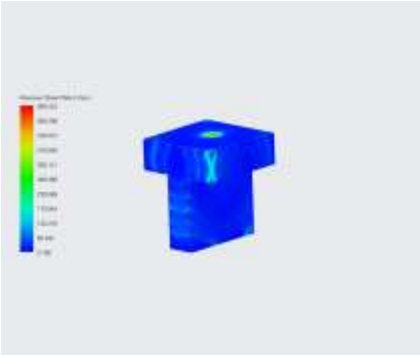
Center temperature is the melt temperature of the middle layer (part line) in the thickness direction at the end of filling. Center temperature is an indicator of the thermal energy supplied to the fresh hot melt. If the center temperature is too low, flow hesitation occurs, which can cause short shot problem.



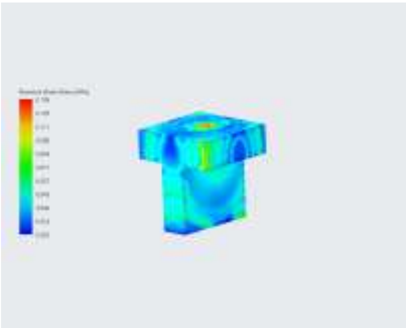
Bulk temperature is a velocity-weighted average temperature of plastic melt across the thickness at the end of filling. The contribution from frozen layer that is stationary is ignored. The effect of heat convection and viscous heating can be displayed from this data. The bulk temperature distribution reflects the trend of the flow path.



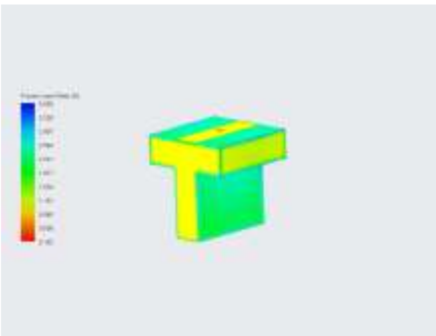
Maximum shear rate result shows the peak value of shear rate at each element during the filling stage.



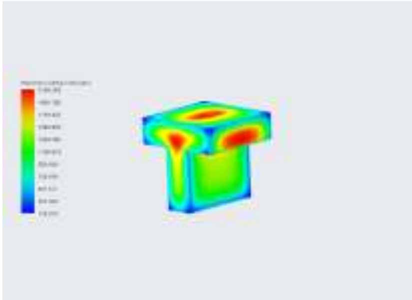
Maximum shear stress result shows the peak value of shear stress at each element during the filling stage.



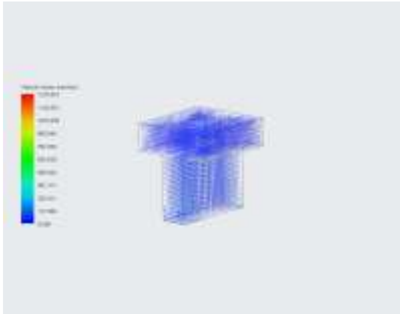
A frozen layer is formed near the cavity surface because of solidification, which is caused by cooling. The frozen ratio gradually increases with time. An increase in frozen layer ratio reduces the cross-section along the flow path and thus increases the flow resistance and sprue pressure. The frozen layer ratio also affects the residual stress and the flow-induced orientation.



This result shows the maximum cooling time in the thickness direction at the end of filling. The cooling time is the time from the end of packing to the instant when the molded part cools to the ejection temperature.



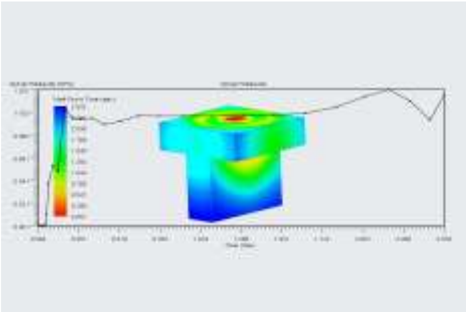
Velocity vector is the vector plot of the velocity vector at the end of filling.



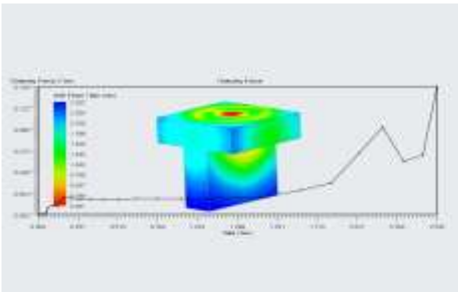
This result shows the maximum volume shrinkage across the part thickness at the end of filling. A high positive value represents high volume shrinkage, which may cause sink mark or void.

This result shows the contribution of each gate. The region in the same color represents that it was filled by the plastic from the same gate. The percentage of each color indicates its percentage of volume compared with the cavity. This result help determine whether the model was filled by balanced flow pattern.

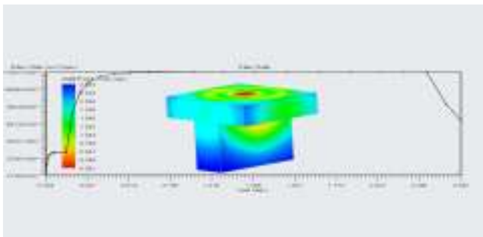
This result shows the plot of sprue pressure versus filling time. You can use this result to look for any unusual sprue pressure rise during filling. If the resulting sprue pressure curve stays at the maximum allowed injection pressure, hesitation or even short shot might occur.



This result shows a plot of the clamping force versus the filling time. The value is the required clamping force during the molding process instead of the force that molding machine outputs. You can use this result to identify the flash problem. If the calculated clamping force is larger than 70 percent of the machine maximum clamping force, there is a chance of plastic melt being squeezed outside the cavity and causing flash.



This result shows the plot of flow rate at the sprue versus filling time. In most cases, the first stage of filling is controlled by the flow rate set by the machine operator. Therefore, in this result, the flow rate usually stays at the value set in the process condition. If the resulting flow rate appears otherwise, check if the maximum allowed injection pressure is too low.



Problem Description:

- Air traps found inside the cavity. This may cause voids or surface defect.

Solution:

- Increase the filling time.
- Change the gate location.
- Reduce the injection speed.
- Modify the size or location of the vents.

CONCLUSION

Injection molding is a complex process with many complex problems. In addition to the aesthetic properties of the product various physical conditions of the molding process should be looked into to prevent defects that can impact the quality of the product. In the present paper the study is focused on prevention of defects arising due to the process parameters which are often encountered in the manufacturing of ball point pen. The major process parameters which are analyzed are clamping tonnage, cooling time, pressure drop and heat transfer rate. Different factors cause various defects in the products like flash, burn marks, short shot, shrinkage, weld line, warpage and sink mark during the manufacturing process.

REFERENCES

1. Chao Chyun An, Ren Haw Chen, The experimental study on the defects occurrence of SL mold in injection molding, journal of materials processing technology , vol-20I, (2008) 706-709.

2. Lei Xie, Gerhard Ziegmann, Mechanical properties of the weld line defect in micro injection molding for various Nano filled polypropylene

composites, Journal of Alloys and Compounds, vol509, (2011) 226-233.

3. Mehdi Moayyedian et al, The analysis of short shot possibility in injection molding process, Int. J Adv. Manufacturing Technology, (2017) 91:3977–3989

4. Daniele Annicchiarico et al, A methodology for shrinkage measurement in micro-injection molding, Polymer Testing, vol- 32 (2013) 769–777

5. Shih-Chih Nian et al, Warpage control of thin-walled injection molding using local mold temperatures, International Communications in Heat and Mass Transfer, vol-61 (2015) 102–110

6. Xianjun Sun et al, A new characterizing method for warpage measurement of injection-molded thermoplastics, Polymer Testing, https://doi.org/10.1016/j.polymertesting.2019.03.024

7. Yu Wang et al, Spiral and conformal cooling in plastic injection molding, Computer-Aided Design, http://dx.doi.org/10.1016/j.cad.2014.11.012

8. Ming-Shyan Huang et al, Influence of clamping force on tie-bar elongation, mold separation, and part dimensions in injection molding, Precision Engineering, http://dx.doi.org/10.1016/j

9. Ming-Shyan Huang and Cheng-You Lin, A novel clamping force searching method based on sensing tie bar elongation for injection molding, International Journal of Heat and Mass Transfer, 109 (2017) 223-230.

10. Xundao Zhou et al, Feature extraction and physical interpretation of melt pressure during injection molding process, Journal of Materials Processing Technology, S0924-0136(2018)30222-X

11. Abohashima HS et al, Minimization of Defects Percentage in Injection Molding Process using Design of Experiment and Taguchi Approach, Industrial Engineering & Management, ISSN, 10.4172/2169-0316.1000179

12. Marco Sorgato et al, Tribological effects of mold surface coatings during ejection in micro injection molding, Journal of Manufacturing Processes, vol-36 (2018) 51–59

13. K Prashanth Reddy and Bhramara Panitapu, High thermal conductivity mold insert materials for cooling time reduction in thermoplastic injection molds, Materials Today: Proceedings, 4 (2017) 519-526

14. N.Crisan et. al, Tribological assessment of the interface injection mold plastic part, Tribology International,https://doi.org/10.1016/j.triboint.2016.04.015

15. MohdJamsheed et. al, Design and Analysis of Plastic Injection Mould for CAM BUSH with Submarine Gate, Materials Today: Proceedings, 2 (2015) 2s083 – 2093

16. Azieatul Azrin and M. Azuddin, Study of the effects of injection molding parameters on weld line formation, Procedia Engineering, 184 (2017) 663-672

17. Lakshmana Naik T K et. al, A study on reducing the sink mark in plastic injection moulding – Taguchi Technique, International Journal of Engineering Research and Development , vol 10, Issue 3 (march 2014), PP 40-43