IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 12, December 2016
ISSN (Online) 2348 – 7968 | Impact Factor (2015) - 4.332
www.ijiset.com

Process Parameter Optimization of Injection Mold Using Concurrent Approach

Jithin k[1], Kannakumar k[2]

[1] ME student ,Manufacturing Engg, Shree Venkateswara Hi-Tech Engineering College, gobi, erode(india)

[2] Assistant Professor, Mechanical Engg, Shree Venkateswara Hi-Tech Engineering College, gobi, erode(india)

Abstract

The injection molding is an important manufacturing process in the production of plastic parts. The design of injection mold is critically important to product quality and effective product processing. CAE tools have been used in the design stage to reduce the losses to obtain the shortened lead time, high quality and achieving low cost of the mold .The design of an injection mold process involves the simultaneous consideration of plastic part design, mold design & Injection molding machine selection, production scheduling& cost as early as possible in the design stages. Concurrent approach effectively eliminates the use of trial and error method by validating and optimizing the design the design of plastic before production.

Keywords: *Injection molding, Mold design, mold flow analysis, mold flow simulation.*

1. Introduction

Injection molding is the effective method for mass production of plastic parts with complicated forms and high precision. In this method, a high pressure fluid polymer is injected to the impression formed in the mold by using injection molding machine. The injection cycle begins when the mold closes, followed by the injection of polymer into the mold cavity. Once the cavity is filled, a holding pressure is maintained to compensate for material shrinkage. Once the part is sufficiently cool, the mold opens and the part is ejected.

CAE technology use injection molding for product design, mold design and mold flow analysis. Firstly the specified product is designed using 3d mechanical software, prepare its core, cavity and entire mold using mold tools. After the product and mold preparation import the models into mold flow software for analysis and simulation. It can forecast the result of injection molding for plastic in designed cavity, find the different problems in them and modify. It proved that numerical simulation technology in injection molding played a key role in quickening new plastic product development, increasing plastic quality and reducing the cost.

2. Literature review

Erzurumlu et al [1] investigated on minimization of the warpage and sink index in terms of process parameters of the plastic parts have different rib cross-section types, and rib layout angle using Taguchi optimization method. Confirmation analysis test with the optimal levels of process parameters are carried out in order to demonstrate the goodness of Taguchi method.

Jianga et al [2] investigated an implicit control volume finite element method for simulation of injection moulding, the time steps were controlled for both flow and thermal simulation by local flow information, and then the computing complexity analysis was conducted. The implicit scheme was based on updating the melt–air interface.

Hassan et al [3] To investigate the effect of the gate location on the cooling of polymer by injection moulding, have carried out a full three dimensional time-dependent analysis for a mould with cuboids-shape cavity having two different thicknesses. The cooling of the polymer material also analysed.

Chen et al [4] previously, production engineers used either trial-and-error method or Taguchi's parameter design method proposed an approach in a soft computing technic for the process parameter optimization of multiple-input multiple-output (MIMO) plastic injection moulding process

Rajalingam et al [5] investigated the process parameters which will affect the shrinkage defect on a plastic cell phone shell component which are Mould temperature, injection pressure and screw rotation speed. The Design of Experimental (DOE) approach was used to investigate and identify the optimal moulding process parameters setting. Statistical results and analysis are used to provide better interpretation of the experiment.

3. Methodology

3.1 model of tail lamp lens

A 3d model of tail lamp lens is created in Catia v5.material is chosen as PMMA. Volume: 15.86 (cm3), Weight: 19.26 (G),Size :X: 120.08 (mm),Y: 75.00 (mm), Z: 80.00 (mm)

Fig. 1 tail lamp lens 3d model

3.2 model of tail lamp lens mold

A 3d model of tail lamp lens mold is created in catia v5.two cavity mold mold is chosen

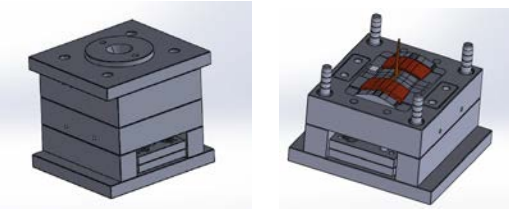

Fig. 2 Tail lamp lens 2 cavity injection mold

3.3 model of tail lamp lens with proper feeding system (runner and gate)

Fig. 3 Tail lamp lens with proper feeding system

3.4 analysis of tail lamp lens mold by solidworks plastic

Importing the model into solidworks plastic. Mesh the model. Input component material as pmma, mold material as steel420.set machine parameters melt temperature 230^{o}c,injection pressure100mpa.set boundary conditions and injection location. Run flow analysis

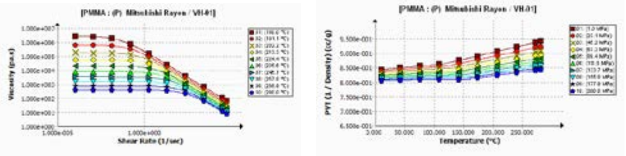

Fig. 4 Pmma properties

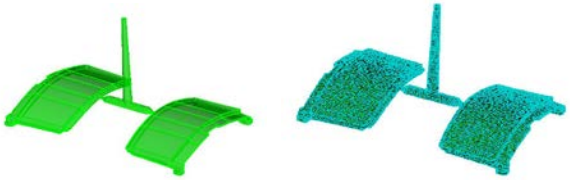

Fig. a importing model

Fig. b meshing model

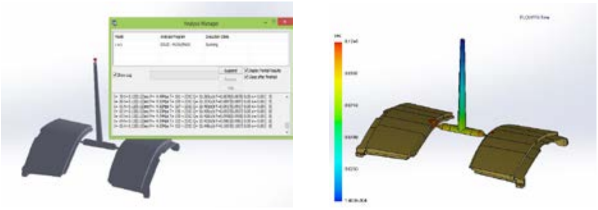

Fig. c set parameters

Fig. d start run

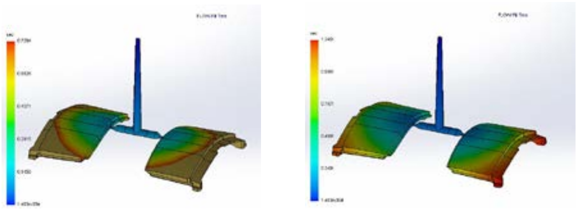

Fig. e filling start

Fig. f finish analysis.

Fig. 5 mold filling analysis of tail lamp lens.

3. Results

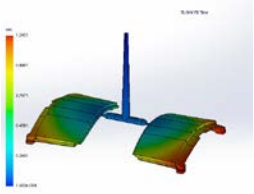

Fig.6 flow vs time

Fig.7 flow vs pressure

Fig.8 flow vs ease of fill

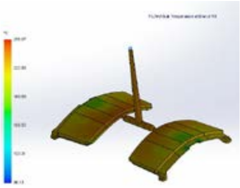

Fig.9 flow vs temperature

X-dir. Clamping Force= 2.1423 Tonne
Y-dir. Clamping Force= 3.3604 Tonne
Z-dir. Clamping Force= 5.9054 Tonne
Requiring injection pressure= 58.5092 Mpa
Max. bulk temperature= 256.2212^{o}C
Max. shear stress= 11.7596 Mpa (1710.0000 psi)
Filling Time = 1.21 sec
Pressure Holding Time = 4.05 sec

4. Conclusions

This part can be successfully filled with an injection pressure of 58.5 MPa (8487.93 psi). The injection pressure required to fill is less than 66% of the maximum injection pressure limit specified for this analysis, which means under specified limit.

The increment of gates reduces the filling time, but it increases the mold complexity in this model, so one gate location is chosed.it reduces the wastage of materials.

Since the flow front temperature has exceeded the starting melt temperature by more than 30° C (86° F), the risk of plastics material degradation is high. This can lead to discoloration and a significant decrease in the physical properties of the molded parts. Steps should be taken to minimize shear heating effects and achieve a more uniform temperature distribution. These include increasing part, runner and gate thickness, increasing mold and melt temperatures and decreasing the plastics volumetric flow rate. It was revealed that the potential problems may be avoided by using analysis function of Moldflow software. During the mould designing process, adopting CAE technique, repeat die repair and trial production times was reduced, high plastic quality was obtained, the period of die designing was shorten.

References

[1] Chi-Huang Lu, Ching-Chih Tsai “Adaptive decoupling predictive temperature control for an extrusion barrel in a plastic injection moulding process” IEEE transactions on industrial electronics, vol 48,october 2001 pp 968-975
[2] ShunliangJianga,b,ZhiguoWanga,GoufaZhoub,Weim inYangc“An implicit control-volume finite element method and its time step strategies for injection moulding simulation”Computers and Chemical Engineeringvol 31, December 2006 pp 1407-1418
[3] Hamdy Hassan, Nicolas Regnier, Guy Defaye“ A 3D study on the effect of gate location on the cooling of polymer by injection moulding” International Journal of Heat and Fluid Flowvol 30, June 2009 pp 1218-1229
[4] Wen-Chin Chen a, Gong-LoungFub,c, Pei-HaoTaib, Wei-Jaw Deng d“Process parameter optimization for MIMO plastic injection moulding via soft computing”Expert Systems with Applications vol 36, 2009 pp1114–1122
[5] S.Rajalingam, Awang Bono and Jumat bin Sulaiman“A statistical experimental study on shrinkage of injection-moulded part” International Journal of Humanities and Management Sciences (IJHMS) Volume 1, Issue 1, 2013 pp 2320–4044