Enhancement and Improvement the Quality of Plastic Products by Estimating and Controlling the Effecting Design Parameters on the Mold

Hani Mizhir Magid, Malik N. Hawas and Badr Kamoon
Al-Furat Al-Awsat Technical University, Kufa, Iraq

Abstract: Injection molding process is a method for mass production which includes melting the polymer into the mold by using high pressure and special tools. This process includes different design parameters that may influence the, accuracy, dimensional stability and finally the product quality. The objective of this study is to analyze and review these parameters and identify the one which mostly affect the product quality. Therefore, various techniques are employed to attain the results. The first step involved the selection of the plastic part, then complete design for the mold by using AutoCAD Software. CATIA Software also used in the modeling stage, followed by the product analysis using the Moldflow Insight and the Master Cam programming. In order to fabricate the mold, CNC machine was employed and finally, the injection molding machine was used to test the mold. Based on the comparison of the previous results, recommendations can be derived to assist researchers in solving problems and overcoming the obstacles in improving the product quality. For instance, controlling the cavity pressure during the injection process will give high dimensional stability with no flash during production process. Moreover, a perfect runner design will reduce the possibility of formation a parting line in a product. Also, it can be concluded that when the contact force between the product and the cavity through a suitable design is decreased, the required ejection force to remove the product is also decreased and maintain the product from any distortion with fine quality.

Key words: Mold, parameter, modeling, plastic, quality, fine quality

INTRODUCTION

The popular technique for producing plastic products nowadays is by using the plastic injection molding process. Also, the demand for small and cheap polymer products is growing rapidly. Injection molding techniques is one of the possible solutions to produce these parts (Kennedy, 2008). It can be used to produce complex shapes with micro scale dimensions in high volumes and short cycles. Injection molding is a cyclic process where the polymer is heated above its melting point and injected into a mold with a temperature far below the melting point and due to this temperature difference the polymer will solidify. In order to compensate the volume loss during this solidification process, polymer is injected under high pressure. After fully solidifying, the mold is opened and the product is taken out of the mound and the cycle is restarted (Baltussen, 2006).

When the parts are short and thin, the melt cannot solidify prematurely and the part may be rejected if not have suitable cooling. Three processing parameters influence this effect which are the mold temperature, the resin temperature and the injection velocity (Tor *et al.*, 2000). Increase both the mold and resin temperatures leads to better molding properties. However in most cases, they will lead to low cycle times and possible material degradation, respectively. Polymer melts have very distinct flow characteristics because the chain-like structure of polymer molecules. If these molecules are close to each other as in melts, the movement of the molecules is constrained by the surrounding molecules. This leads to interesting how-effects such as time dependent behavior of stress, shear-thinning and normal stresses in a shear flow. Materials which show these effects are called viscoelastic materials. Viscoelastic materials behave like a liquid at low deformation rates and as an elastic solid at high deformation rates (Villalon, 2005).

Need for the study: This study was done to determine and analyze the most effect parameters on the product quality in injection molding process and to gain a better understanding about the relationship between mold specifications and the quality. Investigating these effects is depending on scientific analysis and smart design process. Also, it's constrained on examining the effects of molding ejection system on dimensions stability.

Corresponding Author: Hani Mizhir Magid, Al-Furat Al-Awsat Technical University, Kufa, Iraq

Main objectives: This study was done in order to:

- Specify how to avoid the deformation and over flow by controlling the injection pressure which required for filling the cavity
- Determine the durability of selected tools.
- Modeling and analysis this manufacturing process and expand the knowledge for this types of process

MATERIALS AND METHODS

The following sections are used for detailed description of the design, analysis and the validation of this research.

Part design process: The research has employed a method that is able to reduce the errors as much as possible. This method involves a measurement for all dimensions with tolerances and the shrinkage value after the part material is specified. To design any plastic part as in Fig. 1 and 2 some rules must follows in order to ensure better mold design, these rules include:

- Simplified the design to reduce tooling cost
- Using a suitable draft angle to remove the part without deformation
- The minimum wall thickness should be ~0.025 in
- Sharp corners should be avoided and hide weld lines

The next important step was to use AutoCAD Software for drafting a complete mold design to build the mold. A design for two cavity mold and an assembly drawing were drafted to show the quantity and the positions of these parts. In order to secure the accuracy,

each part in the assembly drawing has an individual drawing that illustrates all the necessary particulars for fabricating process (Kahya, 2009). The mold details are shown in Fig. 3.

Modeling and fabrication process: One of the most important steps of the system development process is the definition of the set input and output parameters. These parameters includes mass of the product, overall dimensions of the product, number of cavities, overall size of the mold, type of runner system, heat treatment and the complexity of the mold surface.

Methodology process includes modeling process by using (CATIA) Software. For that, modeling all the detail of parts drawing including the assembly drawing as in Fig. 4. This modeling is important to ensure the design credibility and also to ensure the ability of assembling all these parts with each other's after fabrication process and finally the ability of this design to do the main function during the test by injection molding machine. The modeling process include modeling all items using the
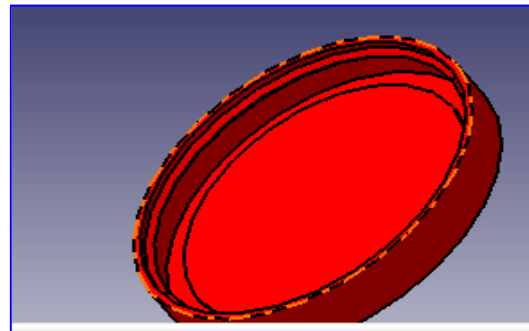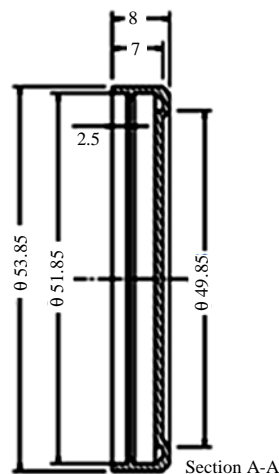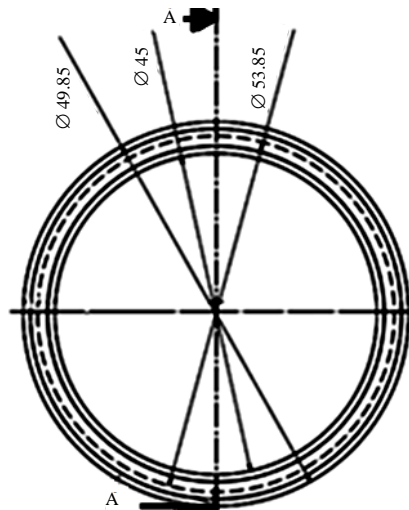

Fig. 1: Plastic cover

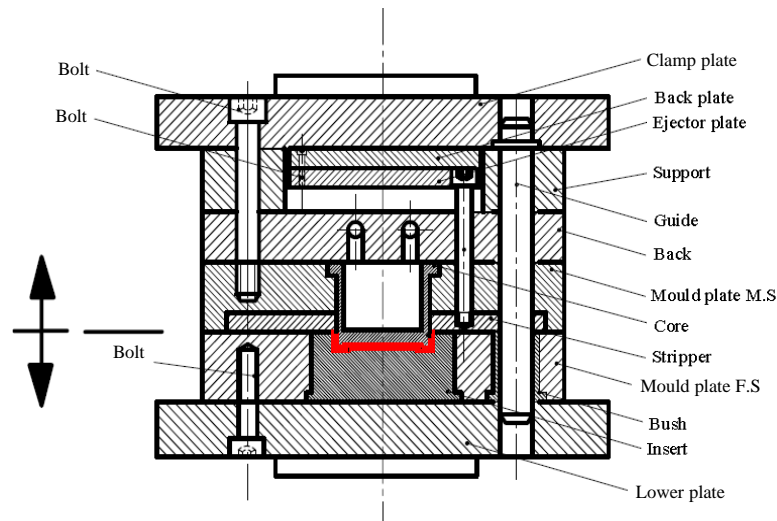
Fig. 2: Part drawing

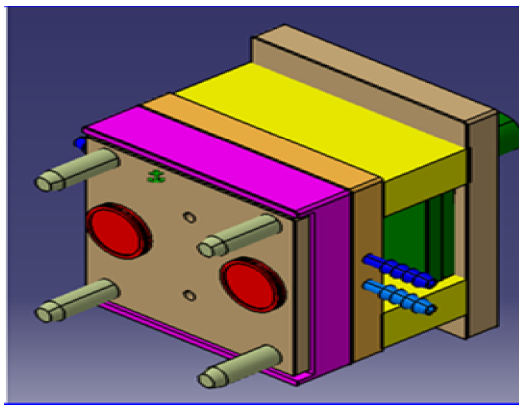Fig. 3: Cross sectional view of an assembly mold

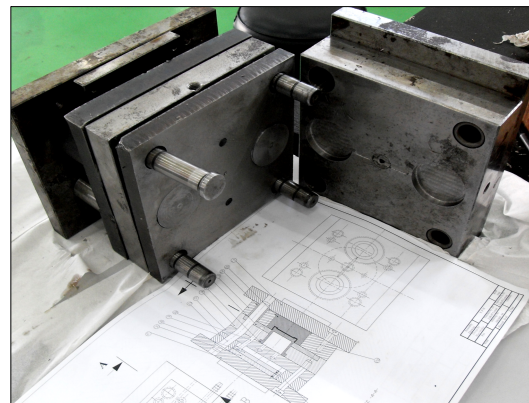
Fig. 4: Modeling for moving part of assembly

Fig. 5: Mold assembly

correct dimensions and assembly drawing modeling. From the options facilities, it's possible to collect all the required information's about the weight, volume and area for all the parts.

The mold base structure consist of many pieces of steel material with (7.86) kg/m³ in density linked together to form the assembly mold. The total dimensions are (200×200×230) mm and the total weight of the mold = (200×200×230×7.86/100000) kg. Figure 5 shows the assembly mold.

Calculations of injection parameters: The raw material of the plastic part is (ABS) has (1.2) kg/m³ in density (2)% shrinkage and (1) mm wall thickness. From part properties in (CATIA) some important data were obtained, like part mass, density and surface area. The calculated dimension after shrinkage:

$$(L\delta) = Lo + (Lo \times 0.02)$$

Where:

$L\bullet$ = The final length after injection
Lo = The primary length

The calculations of these parameters before the injection process is important to avoid any defects or damage in the mold components and to prevent any contact problems:

$$F_{max} = P_{max} {}^* A_{max} \tag{1}$$

Where:

P_{max} = A cavity pressure
F_{max} = A cavity force
A_{max} = Maximum projected area (injection)

The clamping force of the machine during the test should be more than this value. The ejection force is that force necessary to eject the product out of a mold:

Table 1: Input parameters through mold flow

Parameters	Recommended values
Mold temperature	25°C
Injection speed (average)	15 m/sec
Melt temperature	240°C
Nozzle holding pressure	15 MPa
Injection pressure (Average)	80 MPa
Injection time+curing time	15 sec
Mass (product+runner)	30 g
Clamping force	40 ton

$$F = [E*A*m*\ d\ t]/d\ [(d/2t)*(d/4tn)] \qquad (2)$$

Where:

F = A Force which overcome the resistance between mold and product (KN)

E = Modulus of polymer (N/mm²)

A = A contact surface area (mm²)

m = A coefficient of friction

d = A diameter of circle equal to the perimeter of the molding (mm)

t = A wall thickness of the part (mm)

n = A poisson's ratio

During filling, packing and cooling, the material exposes to a complex thermo mechanical stresses which leads to changes in local specific volume. These stresses will develop in the part during solidification and further shrinkage may also, occur during cooling after ejection process. The non-uniform shrinkage behavior will result in a degree of warpage. The filling time during injection process can be calculated by adding; Twice the mold open/close time (2M)+injection time (T)+cooling time (C)+ejection time (E).Where (T) is found by dividing mold cavity size (mm³) on material flow rate (mm³/sec):

$$Total\ time\ =\ 2M+T+C+E \qquad (3)$$

Prediction the total shot numbers (Mold Life ML) of the mold before failure is:

$$ML = f\ (Gi,\ \theta s,\ LT,\ TC,\ CT) \qquad (4)$$

Where:

LT = Layer Thickness

TC = Thermal Cure

Gi = Part dimensions

•s = Draft angle

Parameters setting and analysis by moldflow: To decrease the mistakes, it is important to estimate all the required information's for actual run theoretically using analyzing process by Moldflow to show how much it's near from the actual; also, it's recommended to make the actual test relative to the input values used in Moldflow analysis and compare this test with actual findings. The input parameters used for analysis by Moldflow are shown in Table 1.

RESULTS AND DISCUSSION

Therotical and practical results: There are two types of runs by Moldflow. The first one is the automatic run and the second is data input run. The first one is done by import the part from CATIA in (Stl.) format and then specified the injection point of the part after mesh the part with selecting the type of material, so, the analysis will run and gives the average results values. The second type of run depends on input all parameters values that getting from calculations and make the analysis to get the final results report. The first run will help for get some unknown values which there is no enough information's to calculate it. So, based on the calculations listed before and the test that was done many times by Moldflow with different values of injection pressures and clamping force through each test and melting temperature, the charts in the Fig. 6 a and b shows, respectively the pressure-time analysis and temperature distribution by Moldflow Software.

The experimental results were obtained from testing the fabricated mold by using injection machine. Figure 7 illustrates and shows some products after testing the mold practically by using the injection machine.

In this test, the adjustment and setting of the machine parameters is to obey the required calculations for the parameters that need to check. For this reasons the injection machine was set on different values of injection pressure, melting temperature, filling time and clamping force are used during the experiments. These values will change according to the requirements that need for check to find the high affects parameters on the quality. Figure 8 shows the relation between the wall thickness and mold temperature variance.

If we want to discuss and compare between the theoretical and practical results, it's important to say that; the injection pressure is the important that parameters which gives high effects on the product quality. The accurate estimation of the pressure through the calculations will helps to decrease the probability of weld line formation in the product. The second important parameters that effect directly on the quality is the feed system (sprue bush, runner, gate). The shrinkage will be uniform whenever the flow is smooth with pressure balance and the entrance to the cavity is homogenous with suitable melt temperature.

The other parameter is the ejection system where the ejection method (by pneumatic, stripper plate or by vacuum) is very important to avoid any deformations in the product and to hide any sign for the ejectors on the product .The calculation of the required ejection force before injection process also important in the machine selection.
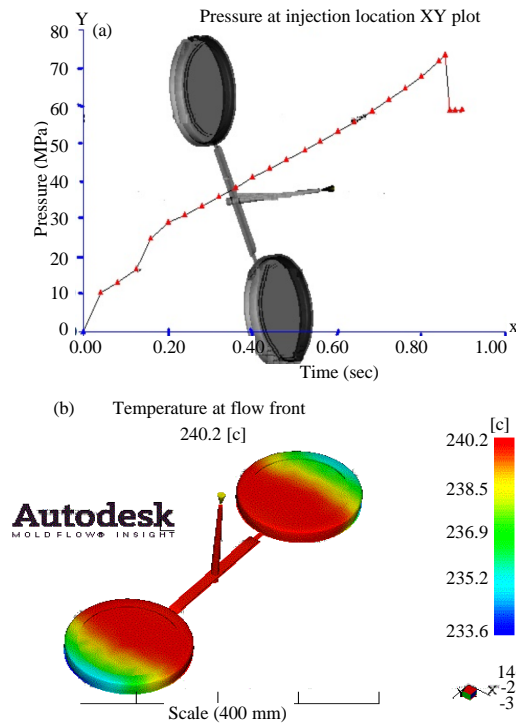
Fig. 6: a) Pressure-time diagram and b) Temperature distribution

Fig. 7: Some plastic cover products with their nozzle experimentally
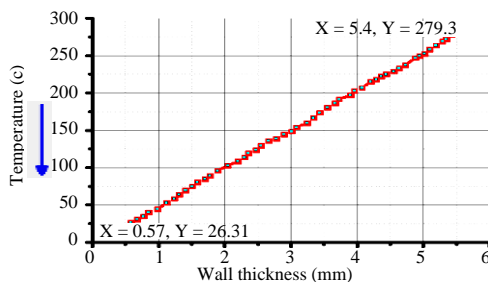

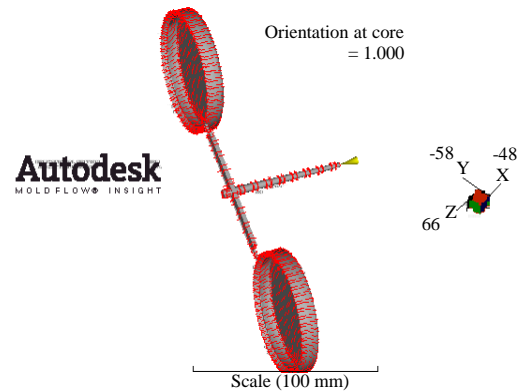Fig. 8: Relation between wall thickness and cooling efficiency

Fig. 9: Core orientation flow

Analysis: The main purpose of analysis in this research is to get advice and early prediction about how to solve or avoid the design problems before it's happened. For example, the ability to make a good design for the feed system, cooling and suitable injection pressure is possible. Also, exposure the mold to different melt temperature and difference pressures will give the designer a good ability to avoid any design mistakes, like choosing the gate shape, gate dimensions and control other parameters.

To make sure that the flow distribution in the cavity is uniform, it's important to show the movement and orientation of the flow during the process. There are two orientation flows, first one is the skin flow and the second is the core orientation. Figure 9 illustrate the core orientation flow.

Summary: The mold design must follow the machine specification to get correct dimensions. The ability to produce high quality product is depend basically on the accuracy of the mold design, so all the required information's should be available before embarking the design. This information's like machine specifications, material specifications, availability of raw materials and also it's necessary to conduct with many software to decrease the mistakes. From the problems happened during fabrication, it's important to say that a good planning, good designing and the ability of using the available software which conducting with the machining tools, will assure the quality. The requirements of fabricating and design will enable the designer to concentrate on the quality. There are another important things should be available in designing and fabricating process, that is the measurement and inspection because without inspection some parts cannot fit with each other's during the assembly process.

CONCLUSION

There are some deviations between the practical and theoretical results because the practical results coming from setting the parameters during injection process continuously. The following conclusions are building up.

Product dimensions can be controlled by adjusting the injection pressure and melting temperature as ideal conditions. Cavity pressure is that parameter which controls the quality of the surface in touche with the cavity and controls the flash. The core orientation is important for giving a uniform wall thickness for the product, also, the skin orientation has the same importance

Mold temperature and cooling effectiveness is important to decrease the cycle time and give high dimensional stability during the ejection. The parameters that control on the surface quality inside the cavity are the pressure inside the cavity, profile shape of the cavity itself and melting temperature of the plastic material. A suitable design and selection for ejection system is important to avoid any distortion.

REFERENCES

Baltussen, M.G.H.M., 2006. Viscoelastic flow effects in high speed injection moulding. Master Thesis, Tokyo University of Foreign Studies, IIS Todai, Tokyo, Japan.

Kahya, E., 2009. The effects of job performance on effectiveness. Intl. J. Ind. Ergon., 39: 96-104.

Kennedy, P.K., 2008. Practical and Scientific Aspects of Injection Molding Simulation. Eindhoven University of Technology, Eindhoven, Netherlands isBN:9789038612751, Pages: 143.

Tor, S.B., S.G. Lee and S.C.Y. Hian, 2000. A two-stage collapsible core for injection Moulded plastic parts with internal undercuts. Intl. J. Mach. Tools Manuf., 40: 1215-1233.

Villalon, A.V., 2005. Electron beam fabrication of injection mold tooling with conformal cooling channels. M.Sc. Thesis, North Carolina State University.