Using Injection Molding Process for Manufacturing a HDPE Mini Wheel

Thi Truc-Ngan Huynh*
Department of Mechanical Engineering,
National Kaohsiung University of Applied Sciences Kaohsiung, Taiwan, R.O.C.
*E-mail: nganhuynhspt39@gmail.com

Abstract

Polymer industry is more and more important with many practical applications, especially plastic mini wheels. With many various shapes and sizes of mini wheels used widely in the families and companies. Moreover, injection molding is the most effectively used process to manufacturing plastic products. In this paper, a combining design and three kinds of software were used to fabricate a prototype of mini wheel. First of all, a model of mini wheel was created by using Pro-E software and then Ansys software used to check sustainable loading capacity of mini wheel. Next, a cooling system was built through using Creo Prametric 1.0 software. The final, Moldflow-CAE software used to analyze the fill time of runner system, the temperature, the pressure, and defects in designing of mini wheel mold. The results showed that the injection plastic flow of the runner with two gates on one product will be better than that of one gate or three gates because it will be easy to move plastic to cavity of product. Besides, Moldflow software can identify some parameters such as the fill time, the pressure and the others that effect on injection molding process. Kinematic and static equations were formulated. The FEA results are a good agreement with the PRBM. The optimal results indicated that the input horizontal force of 0.8896 $N$ and rotary angular of 0.4363 $\text{radian}$. An input rotational angle is the most significant parameter with contribution of 44.2544 %. The fatigue analysis determined that the mechanism could reach approximately one million life cycles before failure.

Keywords: Plastic mini wheel; Injection molding process; Ansys; Moldflow

1. Introduction

Nowadays, plastic injection molding is one of the most important polymer processing operations in the plastic industry. A created product is always depends on many related factors and usually appears many fault on product. Tang et al.[1] used Taguchi method in the design of plastic injection mould for reducing warpage. They determine some parameter units that could be suitable for reducing warpage such as melt temperature (2400$^\circ$C), filling time (0.5 s), packing pressure (90 %) and packing time (0.6 s). Besides, Hassan et al.[2] indicated that the cooling system leads to minimum cooling time is not achieving uniform cooling throughout the mould.

There are many kinds of different applied-and colored polymer materials. Material becomes more important in many manufacturing process. High-density polyethylene (HDPE) is the most suitable material for purposes such as higher specific strength, harder, more opaque, can withstand somewhat higher temperatures, suitable for all kinds of terrains, can stand every acids, bases, deformable forces of loading capacity, and not effect with sunlight. Besides, there are some parameters of HDPE can support the manufacturing process such as density (960 Kg/m$^3$), Young’s modulus (700 Mpa), Poisson’ ratio (0.4), and tensile ultimate strength (50 Mpa). In addition, there are some notes about some manufacturing conditions with HDPE: not necessary dry if having good preserves, melt...
temperature 2000°C, and cavity temperature 600°C. Therefore, in this study a mini wheel prototype made of HDPE. Although there have been many processes such as forging process, extrusion process to manufacture a mini wheel prototype but injection molding process is the most suitable taken into account because of its advantages with reducing time-costing, easy to process, and productivities.

This study focuses on design and manufacturing of mini plastic wheel. Through Pro-E, a model was created; Creo Prametric 1.0 software was used to build a cooling system. A finite element analysis (FEA) based on Ansys software analyzed the deformation and the stress in a proposed wheel. Next, Moldflow software was used to determine the fill time, the pressure and the others that effect on injection molding process. Finally, a prototype was fabricated by using injection molding process.

2. Design and Modelling

Prototype design consists of very important steps. The first step is design a prototype of mini wheel. The second one is to analyze effective factors in injection molding process. The final one is the manufacturing. With mini-wheel prototype was forced on requirements of company about the sizes and the shape. There are some requirements for designing mini-wheel such as hardness and deformation could be suitable with practical conditions, a sustainable loading capacity is less than 50 kg, a diameter of 66 mm, a thickness of 21 mm, a cylinder diameter of 16 mm, a mass of 31 g, and made of HDPE. Thickness of product is also affects directly to the product quality and its prices. Increasing thickness will lead more cooling time, decreased productivity, and some defects such as bubbles, shrinkage, etc. Therefore, the balance of thickness is extremely important step. With using HDPE, the thickness of designing process is about from 0.9 to 6.4 mm. Combining requirements and using Pro-E software, a model of mini plastic wheel was constructed as in Fig. 1.

![Fig. 1. A 3-D model of mini plastic wheel](image)

2.1. Calculation of the Number of Minimum Mold Cavities

The optimum number of mold cavities depend on the desired productivity. In case study, there are three cases to calculate the number of cavities as follows.

Case 1: Depend on ordered product

\[ n = \frac{L.K.f}{24.3600J_m} \] (1)
where

\( n \): the number of minimum mold cavity

\( L \): ration product in batch of goods.

\( K \): coefficient of waste product \( K = \frac{1}{1-k} \) (k: waste product ratio)

\( t_c \): time of plastic injection cycle

\( t_m \): time of completed batch of goods

Assuming that \( L = 4000 \) pieces, \( k = 1\% \), \( K = 1.01 \), \( t_c = 60 \) s, \( t_m = 2 \) days

\( n = 2 \) mold cavities

**Case 2: Depend on injection power of injection machine**

\[
n = \frac{0.85}{W}
\]

\( n \): the number of minimum mold cavity

\( S \): machine power \((g/time)\)

\( W \): weight of product \((g)\)

Assuming that \( S = 267 \) g, \( W = 31.3 \) g

\( n = 6 \) mold cavities

**Case 3: Depend on plastic productivity of machine**

\[
n = \frac{P}{XW}
\]

\( P \): plastic power of machine \((g/min)\)

\( X \): injection frequency in one min \((times/min)\)

\( W \): weight of product \((g)\)

Assuming that \( P = 1233 \) g/min, \( X = 1 \) times/min, \( W = 31.3 \) g.

\( n = 39 \) mold cavities

After calculating, with \( n = 2 \) is the best choose for medium productivities, \( n = 6 \) is suitable for the larger productivity, and \( n = 39 \) is unreal practice. As a result, the best decision is mold with two cavities.

**2.2. Design of Mold**

In case study, the components of a desired mold were selected according to standard of company FUTABA. To design a mold, the first of all is choosing the number of plates, calculating for the thickness of each plate, the sprue, and the runners. Creo Parametric 1.0 was used to design a desired mold, decode and transfer to CNC machine in the manufacturing process; this process is also called CAD-CAM. Fig. 2 shows a setting apart the mold components.
Runner system

The runner system that is responsible for injecting plastic fluid from the out port into the injection machine, including many components such as sprue, main runner, branch runner, gate, part, and cold slug well. The further detailed components are described in Fig. 3.

Sprue bushing

It is used to connect an injecting nozzle and a plastic runner. Figure 4 illustrates 2-D drawing of a sprue bushing.
It used to get plastic fluid flow into cavity. Trapezoid shape cross section is selected (D = 4.7 mm) as shown in Fig. 5 because it can decrease fill time, and more economical material.

This study considered with three proposals for the following runner system.

**Case one:** Design runner for product with one gate on one product (Fig. 6), its mass is 7.3 g.

**Case two:** Design runner for product with two gates on one product (Fig. 7), its mass is 10.5 g.
Fig. 7. Design runner for product with two gates on one product

Case three: Design runner for product with six gates on one product (Fig. 8), its mass is 26.8 g.

Fig. 8. Design runner for product with six gates on one product

Design of a cooling system

It is also priority problem with purpose is to reduce injection cycle in mold design, improve age of mold and productivity, and avoid some faults of product. A cooling system was built through using Creo Prametric 1.0 software (Fig. 9). Based on thickness of a suggested product in the range from 2 mm to 4 mm, so a diameter of cooling pipe was chosen \( d = 8 \text{ mm} \).

Fig. 9. A cooling system

Bubble Analysis

Analytical results by using Moldflow software indicated that bubbles in runner system like violet marks as given in Fig. 10. As this defect, an optimal venting that composes of escape gates as Fig. 11, was designed to
decrease the number of bubbles in molding. With $d = 0.0254 \text{ mm}$, $D = 0.508 \text{ mm}$, $L = 2.54 \text{ mm}$, $W = 6 \text{ mm}$, a venting system was built through using Creo Prametric 1.0 software in Fig. 12.

Fig. 10. Bubbles in a runner system

Fig. 11. Parameters of vent area escape gates

Fig. 12. Model of venting system

**Push system**

By using Creo Prametric 1.0 software, a push system of a proposed mold was designed, including two parts: main parts and assist parts as in Fig. 13. Applying standard of the remains of mold, a completed pushing system was designed as Fig. 14.
Using Injection Molding Process for Manufacturing a HDPE Mini Wheel

After calculating and selecting for overall parts in a mold, the final step was used Creo Prametric 1.0 software to indicate a complete molding model (Fig. 15). It also utilized to support for a practical manufacturing process later.
3. Analytical Results

3.1 Runner

After designing a runner system, some analytical results were indicated. The result of case one, i.e. one gate as in Fig. 16 indicated that plastic flow can not move symmetrically into each mini wheel part. Therefore, choosing case one will be not suitable for fabricating mini wheel. In case two with two gates (Fig. 17), the plastic fill process is better than that of one gate; but the analytical result of three gates (Fig. 18) is the best plastic fill process. However, case three is not an optimal appropriate option for designing mold because total of runner weights in this case are 26.8 g considered so big and no productivity. So the runner with two gates on one product is the best selection for designing runner system.

Fig. 16. Fill time of one gate on one product

Fig. 17. Fill time of two gates on one product
Fig. 18. Fill time of six gates on one product

Moreover, Moldflow software is an useful tool to determine the pressure at the end of fill and bulk temperature as shown in Figs. 19 and 20 (with maximum pressure of 16.02 MPa and bulk temperature time of 1.276 s).

Fig. 19. Pressure distribution at the end of fill
3.2. Cooling System

Using Moldflow software, the analytical results indicated that time to reach ejection temperature is 64.28 s as given in Fig. 21. Fig. 22 illustrates average temperature to evaluate logical cooling system with maximum temperature of 84.55 °C.
3.3. Sustainable Loading Capacity

In this section, Ansys software was used to check sustainable loading capacity of mini-wheel. By applying static structural model with assuming that each wheel will sustain 50 kg, this work set up many steps through step by step. The use of bearing load on the axis of the wheel with alter loads (2-100 kg), the analytical result showed that maximum total deformation is 0.32176 mm (Fig. 23) and maximum equivalent stress is 47.9579 MPa (Fig. 24) in case of the load 20 kg.

Fig. 23. Total deformation
Fig. 24. Stress distribution

With mass of mini wheel in the range from 0 to 100 kg, all simulations were carried out through Ansys software respectively. The simulated results indicated that as load unit is more and more increase, the deformation and the stress are also increased as in Figs. 25 & 26.

Fig. 25. Diagram of deformation distribution of mini wheel under loading

Fig. 26. Diagram of stress distribution of mini wheel under loading
After FEA based on used commercial software as Ansys software, Creo Prametric 1.0 software, Moldflow software; a model of mini plastic wheel was designed and analyzed. Based on the designed and analytical results above, a prototype will manufacture in next section.

4. Experimental Set-up

Choosing type of machine is always the first step before fabricating any products. In this paper, SHINE WELL W-120B machine (Fig. 27) applied to manufacture a proposed plastic mini wheel with some general information such as: Mold opening stroke 380 mm, suitable mold size 295 x 350 mm, injection pressure 1393 kg/cm², injection rate 131 cm³/sec, injection stroke 200 mm, and clamping force 120 ton, etc. All of components were illustrated as in Fig. 28. Next step, this study used MIKRON UCP 600 CNC 5-axis mill machine for processing some plates (support plate, ejector retaining plate, and ejector plate) of mold as Fig. 29. CNC EMCO Mill 155 machine (Fig. 30) was used to process inserting parts. Then, a perfectly fabricated mold was given as in Fig. 31.
Fig. 28. Components of SHINE WELL W-120B machine

Fig. 29. MIKRON UCP 600 CNC 5 axis mill machine and plates

Fig. 30. CNC EMCO Mill 155 machine and inserting parts
Using Injection Molding Process for Manufacturing a HDPE Mini Wheel

The final step is to fabricate a proposed plastic mini wheel by using SHINE WELL W-120B machine. Through fabricating process can evaluate the quality of product and propose some projects that can improve some molding defects as air burn, non-fill, and weld line. A product of fully-constituted plastic mini wheel was manufactured as indicated in Fig. 32.

5. Conclusion

This paper focuses on design and manufacture of a plastic mini wheel. First of all, Pro-E software was used to create a model of mini wheel; a cooling system was built through using Creo Parametric 1.0 software. Next, FEA based on Ansys software was carried out to test the sustainable loading capacity and deformation distribution of a proposed mini wheel. And then, Mold flow software was used to analyze the fill time of runner system, the temperature, the pressure, and defects in design of mini wheel mold. The end of step, a prototype was manufactured.
After fabricating process, some molding defects were indicated. Therefore, to minimize the molding defects is always priority problem. With bubble case, injection mold process should decrease the lost pressure of screws or the press force. Another shrinkage case that this process can check temperature and increase gate dimension at wall thickness position or design sprue and gate with suitable diameters in plastic fill process, and sometimes can use water to cooling each wall thickness. Thus, improving defects of mold is always important that affect the quality of product and the age of product.

References