Improvement on Injection Molding Technology
射出成型模具技术改良

Student: Josue David Ayala Guido

Advisor: Shu-Hsing Chen

Department of Mechanical Engineering
Kun Shan University
Tainan, Taiwan, R.O.C.
Hands-On Project Report

June, 2016
Project Practice Report Authorization Letter

I/we (the Principal), Josue David Ayala Guide, hereby authorize Library and Information Center of KSU (the Agent) to gain access to our project practice report at Department of Mechanical Engineering at KSU on the second (first-second) semester in academic year of 104.

Report Title: Improvement on Injection Molding Technology

The Principal agrees with not only the Agent on digital reforming the full text for repository but also the users on having the access to the report according to Library Law and Copyright Law of R.O.C.

The statement above is no need for making inalienable agreement and authorization contract. Copyright for the full text is non-exclusive license. The Principal would not get paid for any applications of the full text.

Date of scope for publication (select either and make a check in it):

☐ Immediately open
☐ Open for access after five years
☐ Open for access after three years
☐ Open within the library

Instructor’s Name: Shi-Hsiung Chen

Student’s Name: Josue David Ayala Guide

Student Identity No.: 04010258

Date (Date): 2016年 06月

Kun Shan University
http://www.ksu.edu.tw
# Table of Content

Abstract  

1. Introduction of history of injection molding  
1.1 Introduction of plastics  

2. What is injection molding?  

3. Applications of Injection Molds  

4. Mold and Part Design  

5. Some typical plastics and their uses for injection molding  

6. Advantages and disadvantages of Injection molding  

7. Injection molding design  

8. PowerMILL Simulations for the cavity and core  

9. Conclusion  

10. Glossary of some Plastic Injection Molding Terms  

11. References
Improvement on Injection Molding Technology

Abstract

Since the increasing development of plastic industry is a fact and improvement and creativity are essential to implement better plastic techniques for making complex shapes, injection molding is the most popular for producing plastics products. There are many reasons of which two are listed: high productivity and efficiency to manufacture dozens, hundreds or thousands of pieces with simplest or the most difficult shapes. The main purposes to write this paper are to provide basic information, concepts and give some ideas of how important and at some point necessary comes to use injection molding of thermoplastics. By injection molding, plastic parts are easy to make with the injection mold, fast to produce in relation of the complexity in many or couple seconds, have lower price because thermoplastics are cheaper than metals and other materials, and high performance in engineering and different utilities. The paper begins with the definition of injection molding and brief history of how it has become more important while the industry has progressed. Applications of injection molds, plastics and their uses and examples of polymers best suited for injection molds are also subtitles of this paper. The description of a two plate molds, the importance of injection molding, advantages and disadvantages, applications of this technique and drawings of a two plate mold and glossary of words is the last information in relation with injection molding.

Key words: Plastics, injection molding, molds
1. Introduction of History of Injection Molding

How did the world of injection molded parts begin? In 1868, billiard ball maker Phelan and Collander, John Wesley Hyatt invented a way to make billiard balls by injecting celluloid into a mold. Hyatt improved the celluloid so that it could be processed into a finished form. In 1872 John and his brother Isaiah patented the first injection molding machine. This machine was relatively simple compared to the machines used by today's injection molding companies. It contained a basic plunger to inject the plastic into a mold through a heated cylinder. The industry progressed slowly over the years producing injection molded products such as plastic collar stays, buttons, and hair combs. In the 1940's the concept of injection molds grew in popularity. This is because World War II created a huge demand for inexpensive, mass-produced products. In 1946, James Hendry built the first screw injection molding machine, revolutionizing the plastics industry with an auger design to replace Hyatt's plunger. The auger is placed inside the cylinder and mixes the injection molded material before pushing forward and injecting the material into the mold. This allowed colored plastic or recycled plastic to be added to the virgin material and mixed thoroughly before being injected. Today screw injection molding machines account for 95% of all injection machines companies. The industry of injection molds has evolved over the years, from producing combs and buttons to a diverse array of custom injection molded products for the following industries: medical, aerospace, consumer, toys, plumbing, packaging, automotive, and construction.

1.1 Introduction of Plastics

Plastics are made from polymers and are produced in the temperature at which they become plastic in order to obtain the finished product. Polymer macromolecules are composed of groups of atoms that are repeated more or less ordered along the chain. Macromolecules of interest in terms of technology are those whose molecular weight exceeds 10,000. In view of their technical utilization of plastics are classified considering the thermomechanical behavior, which takes into account the variation of deformation of a polymer material under constant load, depending on temperature. Thus there are the following groups of polymers: thermoplastics, elastomers and duroplast. The explosive development of plastics industry due, on one hand the emergence of many new polymers with very different characteristics, on the
other hand improve their processing technologies, the rapid expansion resulted in the last 20 years of plastics applications, the emergence of numerous produced from synthetic polymers or modified natural, artificial metals or other materials shortages, which have invaded the artificial environment evolving lives, decisively influencing socio-economic development. Plastics are some of the most important and widely used materials in the industrialized world. Defined as solid, synthetic, organic polymer materials, plastics are easily formed into almost any shape desired. They have a wide range of physical properties—strength, rigidity, opacity, color, toughness, hardness, ductility, heat tolerance, thermal and electrical conductivity, etc., and can thus be used in a variety of products and applications. The properties of plastics are so diverse that they can be and are substituted for metals in which case they are called “engineered materials” and, when formed into fibers, substitute for natural fibers like silk and wool, (Figure 2). Plastic products are created by the molding, forming, and shaping of solid or liquid resins. Two types of resins are used in the manufacture of plastics—thermoplastics and thermosets. Thermoplastic resins can be heated and formed repeatedly, but thermoset resins, once heated and formed, cannot be remelted. The process of melting a thermoset resin irreversibly alters the internal linkages of the polymers, making it difficult to recycle products made from the thermoset plastic. In contrast, thermoplastics are generally suitable for recycling. Plastics occupy a very important role although in recent years has reduced oil and gas extraction, primary sources of raw materials used in manufacturing plastics. The first industrial plastics were obtained between 1927-1950, so that in the last 70 years over 80% of surrounding has been replaced by products made of plastic. Cellulose was present on the Earth when there were trees and plants as the main constituent of plant cell walls. Because of this there is no specific date for its discovery, appeared before human birth. Recognition of cellulose as a main constituent of plant cell wall was in 1838 due to findings Anselme Payen French botanist who first isolated from wood pulp. That was understood structure. Currently polysaccharide cellulose is the most widespread in nature and is under the guidance of scientists producing real changes in the processing, use and plant genetics. Although obtaining cellulose derivatives was attributed to Professor Christian Shonbein Swedish, English inventor Alexander Parkes and American entrepreneur John Wesley, basically, the latter is considered the father of the plastics industry. Based on metal injection machine, patented in 1870 by John Smith and Jesse Locke, Hyatt brothers made and patented in 1872 the first machine injected molding. This was the first and most important step in achieving development and plastics processing industry. Since then they have taken many steps, big and small but what is most important is that it has developed plastics processing industry so that
today 80% of products manufactured in the world have incorporated at least one polymeric material landmark, [3]. The highlights of the evolution of obtaining and plastics processing industry are presented in Table 1.

**Table 1. Evolution of obtaining and processing of plastics [3]**

<table>
<thead>
<tr>
<th>No.</th>
<th>Period of time</th>
<th>Author(s)</th>
<th>Contributions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1735-1744</td>
<td>Charles Marie de la Condamine-Franca</td>
<td>He participated from 1735-1744 in a scientific expedition to Peru to measure the terrestrial meridian. Down the Amazon River gum trees found in the heart of tropical forest and noted the remarkable qualities of juice that flowed from it.</td>
</tr>
<tr>
<td>2</td>
<td>1844</td>
<td>Charles Goodyear-USA</td>
<td>Working on a method of treating gum India, sell a few drops of a mixture of gum rubber and sulfur on a hot disk. He obtained the first patent but had to lead a struggle with a large number of counterfeit and succeeds only in 1852.</td>
</tr>
<tr>
<td>3</td>
<td>1860</td>
<td>Charles G. Williams-Anglia</td>
<td>He made the first step in obtaining synthetic rubber separating isoprene</td>
</tr>
<tr>
<td>4</td>
<td>1880, 1884</td>
<td>G. Boncharévet-Franta, Tilden-Anglia</td>
<td>First industrial successes were recorded. Production of synthetic rubber (elastomer) really began during the Second World War to replace natural rubber.</td>
</tr>
<tr>
<td>5</td>
<td>1865</td>
<td>P. Schützenberger, G.V. Namdn</td>
<td>They tried to cellulose esterified with acetic acid.</td>
</tr>
<tr>
<td>6</td>
<td>1873</td>
<td>-</td>
<td>Was finding the first acrylic acid ester.</td>
</tr>
<tr>
<td>7</td>
<td>1898</td>
<td>H. Schnell</td>
<td>It was discovered polycarbonate industrial process for Bayer company who sold the first product on the market as the Makronol name.</td>
</tr>
<tr>
<td>8</td>
<td>1897</td>
<td>Frescman, Grevess</td>
<td>Were prepared for the first time polyurethanes.</td>
</tr>
<tr>
<td>9</td>
<td>1907</td>
<td>Rolud</td>
<td>Begin the first studies on polymethyl methacrylate.</td>
</tr>
<tr>
<td>10</td>
<td>1908</td>
<td>C. F. Crum &amp; L.J. Weber</td>
<td>Patents acetylation of cellulose.</td>
</tr>
<tr>
<td>11</td>
<td>1909</td>
<td>Leo Baskeland -USA</td>
<td>Discover bakelite.</td>
</tr>
<tr>
<td>12</td>
<td>1908</td>
<td>Jacques Brandenberger</td>
<td>Invent manufacture of cellophane, transparent new packaging.</td>
</tr>
<tr>
<td>13</td>
<td>1908</td>
<td>Hermann Staudinger</td>
<td>He realised the first study on polymer synthesis.</td>
</tr>
<tr>
<td>14</td>
<td>1921</td>
<td>Eichengrün &amp; Buchholz</td>
<td>They designed a machine for cellulose acetate injection.</td>
</tr>
<tr>
<td>15</td>
<td>1925</td>
<td>-</td>
<td>Kodak and Rhone Poulenc companies made studies for cellulose production.</td>
</tr>
<tr>
<td>16</td>
<td>1930</td>
<td>Weldon Semon</td>
<td>A new material made from waste. The process transforms PVC found in a plastic flexible and durable.</td>
</tr>
<tr>
<td>17</td>
<td>1933</td>
<td>R.O. Gibson-Anglia</td>
<td>He created a new plastic, polyethylene from ethylene gas.</td>
</tr>
<tr>
<td>18</td>
<td>1933</td>
<td>Wulff-Germania</td>
<td>Put up getting polystyrene.</td>
</tr>
<tr>
<td>19</td>
<td>1933</td>
<td>-</td>
<td>Rohm &amp; Haas Company introduced the first methyl polymethacrylate.</td>
</tr>
<tr>
<td>No.</td>
<td>Period of time</td>
<td>Author(s)</td>
<td>Contributions</td>
</tr>
<tr>
<td>-----</td>
<td>---------------</td>
<td>-----------</td>
<td>---------------</td>
</tr>
<tr>
<td>21</td>
<td>1934</td>
<td>Firma DuPont</td>
<td>Polyvinyl fluoride material was introduced.</td>
</tr>
<tr>
<td>22</td>
<td>1935</td>
<td>Firma Farbenindustrie</td>
<td>Was patented styrene-acrylonitrile material.</td>
</tr>
<tr>
<td>23</td>
<td>1935</td>
<td>Fewett &amp; Gobson-Anglia</td>
<td>They obtained low density polyethylene.</td>
</tr>
<tr>
<td>24</td>
<td>1937</td>
<td>Wallace H. Carothers</td>
<td>Nylon was patented.</td>
</tr>
<tr>
<td>25</td>
<td>1938</td>
<td>Ray J. Plunkett</td>
<td>He realised a successful polymerization of tetrafluoroethylene naming new Teflon material.</td>
</tr>
<tr>
<td>26</td>
<td>1939</td>
<td>-</td>
<td>Industrial fabrication of low density polyethylene.</td>
</tr>
<tr>
<td>27</td>
<td>1939</td>
<td>Firma DuPont-USA</td>
<td>Begin selling nylon.</td>
</tr>
<tr>
<td>28</td>
<td>1940</td>
<td>U.S. Ruben</td>
<td>He produced the first acrylic-nitrite-butadiene-styrene.</td>
</tr>
<tr>
<td>29</td>
<td>1940</td>
<td>Regnault</td>
<td>Industrial production of polyvinyl chloride began.</td>
</tr>
<tr>
<td>30</td>
<td>1941</td>
<td>Corrothers, Whinfield, Dicksen</td>
<td>They started the study of linear polyesters.</td>
</tr>
<tr>
<td>31</td>
<td>1947</td>
<td>Firma ICI</td>
<td>Starts of industrial production of low density polyethylene.</td>
</tr>
<tr>
<td>32</td>
<td>1953-1954</td>
<td>Karl Ziegler</td>
<td>Appears the high density polyethylene.</td>
</tr>
<tr>
<td>33</td>
<td>1954</td>
<td>Giulio Natte-Italia</td>
<td>He obtained new synthetic polypropylene.</td>
</tr>
<tr>
<td>34</td>
<td>1958</td>
<td>Firma DuPont</td>
<td>He produced the first polyacetel.</td>
</tr>
<tr>
<td>35</td>
<td>1960</td>
<td>Firma DuPont</td>
<td>Produced the linear polyethylene.</td>
</tr>
<tr>
<td>36</td>
<td>1961</td>
<td>Firma Penn Salt Chemicals</td>
<td>The first industrial-scale production of polyvinyl fluoride.</td>
</tr>
<tr>
<td>37</td>
<td>1964</td>
<td>Allan Hay</td>
<td>He found polyfenchlenoxidil.</td>
</tr>
<tr>
<td>38</td>
<td>1965</td>
<td>-</td>
<td>It was introduced in manufacturing the polysulphones.</td>
</tr>
<tr>
<td>39</td>
<td>1970</td>
<td>Robert Jarvik &amp; Willem Koff</td>
<td>They made the first artificial heart in plastic, aluminum and synthetic fibers called Dacron.</td>
</tr>
<tr>
<td>40</td>
<td>1970</td>
<td>Firma Celanese</td>
<td>The material released polybutylene.</td>
</tr>
<tr>
<td>41</td>
<td>1970</td>
<td>Joseph L. Wirt</td>
<td>He discovered material called polyetherimides.</td>
</tr>
<tr>
<td>42</td>
<td>1970</td>
<td>-</td>
<td>Liquid crystal polymers have appeared.</td>
</tr>
<tr>
<td>43</td>
<td>1981</td>
<td>Gerard Eleens</td>
<td>He realised the synthetic material family (Pebax) used in the manufacture of sports equipment.</td>
</tr>
<tr>
<td>44</td>
<td>1986</td>
<td>Firma Mitsui Petrochemical- Japonia</td>
<td>Iris found polimetlenpentene.</td>
</tr>
<tr>
<td>45</td>
<td>1991</td>
<td>Firma Solvay</td>
<td>A mixture made from poliarilamide and polypropylene.</td>
</tr>
</tbody>
</table>

**Figure 2. Parts used in machine manufacturing**
2. What is Injection Molding?

Injection molding is a manufacturing process for producing plastic injection molds from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mold cavity by a reciprocating screw or a ram injector, where the injection molded part cools and hardens to the configuration of the mold cavity. After a part is designed, usually by an industrial designer or an engineer, molds are then manufactured by an injection mold company, where it is assigned to a mold maker (or toolmaker). Injection molds are usually constructed using either steel or aluminum, and precision-machined to form the features of the desired parts. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made injection molded items including computer components to outdoor furniture.

![Injection Molding Diagram](source: www.offshoresolutions.com) [7]

*Figure 3. A simple reciprocating screw injection molding machine (source: www.offshoresolutions.com) [7]*
3. Applications of Injection Molds

Plastic injection molding is the preferred process for manufacturing plastic parts. Injection molds are used to create many things such as electronic housings, containers, bottle caps, automotive interiors, pocket combs, and most other plastic products available today. Plastic injection molds are ideal for producing high volumes of plastic parts, due to the ability of making multi-cavity injection molded parts, where multiple parts are made with one cycle. Some advantages of injection molding are high tolerances, repeatability, a wide range of material selection, low labor cost, minimal scrap losses, and little need to finish parts after molding. Some disadvantages of this process include an expensive tooling investment and the need to prototype, as some custom complex parts may encounter problems during the injection molding process such as warp or surface defects. Therefore, injection molded parts must be designed with careful molding consideration.

![Diagram of injection molding process](image)

*Figure 4. Schematic diagram of injection: a - injection molding material; B - solidification and cooling of the melt, c - throwing open the mold and the part from the mold: 1 - mobile shelf, 2 - matrix, 3 - fixed shelf, 4 - nozzle, 5 - cylinder, 6 - body heating, 7 - snail, 8 - feed hopper, 9 - drive system rotating, 10 - operating system in translational motion, 11 - piece injected*
Just as in die casting, the mold is specially made for each part, and the basic elements of each mold are the same, including sprue, gates, runners and vents; in addition, the location of ejection pins is usually specified in the mold design, since there points have visible marks (therefore ejection is usually done from the core side, and is usually mounted into the mold half mounted on the moving platen). The cavity is divided between the two mold halves in such a way that the natural shrinkage of the molding causes the part to stick to the moving half. When the mold opens, the ejector pins push the part out of the mold cavity. There will be more details of molds in the information below.

Two-Plate Mold: This consists of two halves fastened to the two platens of the molding machine's clamping unit. When the clamping unit is opened, the mold halves separate. Molds can contain one multiple cavities to produce one or multiple parts in a single shot (last example in figures 5 below). The parting surface is the surface shared by the two mold halves.
Just as in die casting, the mold is specially made for each part, and the basic elements of each mold are the same, including sprue, gates, runners and vents; in addition, the location of ejection pins is usually specified in the mold design, since there points have visible marks (therefore ejection is usually done from the core side, and is usually mounted into the mold half mounted on the moving platen). The cavity is divided between the two mold halves in such a way that the natural shrinkage of the molding causes the part to stick to the moving half. When the mold opens, the ejector pins push the part out of the mold cavity. There will be more details of molds in the information below.
4. Mold and Part Design

Sprue Bushing: A sprue bushing with a standard 2½° included angle, approximately 42mm taper per meter (0.5 in. taper per foot) should be used. The entrance diameter of the bushing should always be slightly larger than the nozzle exit orifice. To promote a balanced pressure to the runners and cavities, the exit diameter of the sprue bushing should be larger than the diameter of the main runner. Z-pin-type pullers are preferred for easy removal of the sprue.

Runners: In a two-plate mold, full-round runners are preferred because they provide the highest volume-to surface ratio, the least pressure drop and are the easiest to eject from the mold. Depending on the part size and weight, typical full-round runner diameters are 0.6 cm to 1.0 cm (0.25 in. to 0.4 in.). Because of excessive flow restriction, small diameter runners, less than 0.6 cm (0.25 in.) diameter, should be avoided. Excessively large-diameter runners offer little advantage and contribute to longer cycle times and greater material usage. If a three-plate mold is being used, full-round runners are still preferred, but trapezoidal runners can be used.

Figure 6 shows typical relative dimensions of a trapezoidal cross-section runner. The flow through a trapezoidal runner is equivalent to that of the largest circular runner whose cross section can be inscribed within the trapezoid. To maintain pressure and balanced flow during injection into a multiple cavity or multigated mold, the secondary runners should be slightly smaller in cross section than the main runner.

![Figure 6](image_url)

*Figure 6
Relative Dimensions of a Trapezoidal Runner for Use in a Three-Plate Mold.*
Secondary runners should be perpendicular to the main runner, and the runner junction should be vapor-honed to remove burrs and sharp edges, and contain a cold slug well at every turn of direction. Figure 7 shows a properly sized runner system.

In addition to proper runner sizing, the layout of the mold is also an important consideration. A runner system should be designed to give balanced flow to all gates, ideally designed so that the melt reaches all of the gates simultaneously. TPU (thermoplastic polyurethane elastomer) compounds have been molded successfully in hot runner systems. In many cases, hot runner mold design and temperature control have become the most complex part of the molding process. Hot-runner molds have usually been designed originally for plastics other than TPU. In general, those created for PVC would offer the best characteristics for success with TPU.

Cold Slug Wells: During injection, the initial surge of material is generally cool since it has remained dormant in the nozzle while the previous shot was being ejected from the mold. To prevent this cold material from entering the cavity and causing a visual defect, cold slug wells or runoffs should be incorporated into the runner system before material is allowed to enter the cavities. Properly sized runner systems designed for balanced flow which incorporate cold slug wells are shown in Figure 8.
Gates: TPU compounds have been molded satisfactorily through a wide variety of gate designs including fan, lab, edge, submarine and sprue. In general, the gates should have a generous cross-sectional area to allow the material to flow freely with a minimum of pressure loss. The gates should be vapor-honed with all rough edges and sharp corners removed. Figure 9 illustrates several acceptable gate designs with rounded corners for minimum restriction. Tab gates are strongly recommended for the softer TPU grades. They eliminate the distortion in the gate area that commonly occurs with very flexible materials. The use of pinpoint gates and tunnel gates should be restricted to very small parts of a few ounces or less in weight where the flow length from the gate is less than two inches. The land length for gates should always be as short as possible. A good rule of thumb for determining the proper land length is that it should be no greater than one-half the gate thickness.

In multigated cavities, the gate location and number of gates are very important in relation to the appearance and performance of the molded part. Since gate areas are almost always more highly stressed due to orientation, gates should be located in noncritical sections of the part. Gating in thick sections of the part and allowing the material to flow to the thinner sections keep sink marks to a minimum. When gating into a thick section, the flow should be directed toward a cavity wall or deflector pin to break up the melt entering the cavity and to prevent a condition called "worming." Worming is a random pattern of weld lines opposite the gate caused by the rapid cooling of the injection melt. If the design of the part requires a split in the flow front coming from the gate, a weld line will usually result when the flow fronts meet. Care should be taken in designing parts to keep the number of gates to a minimum to
minimize weld lines. Multiple weld lines could detract from the surface appearance and may affect performance.

Figure 9 Gate Designs. [4]

Figure 10 Mold Venting. [4]

Dimensions:

\[ A \in [0.001, 0.002] \]
\[ B \in [0.100, 0.150] \]
\[ C \in [0.500] \]
\[ D \geq 0.400 \]
Mold Shrinkage: Mold design, part design and operating conditions all affect the mold shrinkage value of any thermoplastic material. In cases where very close tolerance must be maintained, it is suggested that a prototype tool be made before building the production tool. Where standard or coarse tolerances are all that is required, the standard mold shrinkage allowance for the particular LSP TPU compounds should be used. It should be noted that post annealing or exposing parts to a point over temperature will increase the mold shrinkage from what is normally expected. This data is presented in Technical Data Sheets of LSP TPU molding compounds.

Vents: Because of the low viscosity of the melt, normal vent depths of 0.05 mm (.002 inches) and greater will allow TPU compounds to flash. Therefore, vents should be cut only after initial trials on the new tool have indicated necessary locations. A vent channel 6.4 mm to 12.7 mm (.25 in. to .50 in.) wide by .03 mm (.001 inch) deep is usually sufficient, See Figure 10. The cooling system is made up of passages in the mold that are connected to an external pump. Water is circulated through them to remove heat from the hot plastic. The air trapped in the cavity passes through the small ejector pin clearances in the mold, and through narrow vents that are machined into the parting surface (typically about 0.03 mm deep and 12 to 25 mm wide), See figure 11.
Figure 11. Cooling system [4]
5. Some Typical Plastics and Their Uses for Injection Molding

Thermoplastics
General properties: low melting point, softer, flexible.
Typical uses: bottles, food wrappers, toys, …
Examples:
  • Polyethylene: packaging, electrical insulation, milk and water bottles, packaging film
  • Polypropylene: carpet fibers, automotive bumpers, microwave containers, prosthetic body parts for disabled people
  • Polyvinyl chloride (PVC): sheathing for electrical cables, floor and wall coverings, siding, credit cards, automobile instrument panels
  • Polystyrene: disposable spoons, forks etc., also used to make Styrofoam™ (soft packaging material)
  • Acrylics (PMMA: polymethyl methacrylate): paints, fake fur, plexiglass
  • Polyamide (nylon): textiles and fabrics, gears, bushing and washers, bearings
  • PET (polyethylene terephthalate): bottles for acidic foods like juices, food trays, mylar tapes
  • PTFE (polytetrafluoroethylene): non-stick coating, Gore-Tex™ (raincoats), dental floss.

The most common methods of processing plastics to manufacture plastic parts are similar to methods we have learnt for metals and glass. These include Extrusion, Injection molding, Blow molding, Casting, etc. Among these, perhaps injection molding is the most significant for local industry – almost all manufacturing companies use parts that are injection molded, whether they make toys, home-appliances, electronics or electrical parts, watches, computers, etc.
Figure 12. Examples of Injection Molding Parts [source: www.ymf.com.hk] [6]

Figure 13. Injection Molding Part designed in Solidworks for the previous two plate mold.
6. Advantages and Disadvantages of Injection Molding

Advantages

- Fast production. Plastics part can be produce amazingly fast in a few seconds.

- Material and color flexibility. The produced part can easily be change color and flexibility there is a tool made.

- Labor costs low. Once all the parameters are set (amount material for the part, temperature of the nozzle, clamping cycle, etc.) injection machines can run automatically, so there is no need of many people for this operations.

- Design flexibility. The amount of flexibility of plastic parts can be limitless, the results of such properties and restricted by the manufacturers.

- Low waste. Majority of plastics are recyclable. The wasted material can be grinded and reused for the same purpose.

Disadvantages

- High Initial tooling cost. It is pretty expensive to start producing plastics parts. Injections molding machines, molds, skills to run it and some other tools are necessary.

- Part design restrictions. To make a very good mold design requires a lot of knowledge and years of experience, there are many requirements to follow for working with different size, materials, temperature and related factors.

- Accurate cost. Some plastics parts are not able to be produced by simple methods. It also has relation with cost, and regards with materials to make the molds could be very expensive. Sometimes mold parts have to be remade.
7. Injection Molding Design

The following drawings are part of the two plate mold. These provide more details of the complete design.

*Figure 14. Section view of cooling system for the cavity (a) wireframe and (b) solid*
Figure 15. Properties of the plastic part: sprue, runner, gates, vents, slug, parts of the cavities, all together.

Mass properties of guitar pick

Output coordinate System: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 3.50 grams

Volume = 3496.23 cubic millimeters

Surface area = 4490.11 square millimeters

Material used for the part designed: Acrylonitrile butadiene styrene (ABS) Properties of ABS:

<table>
<thead>
<tr>
<th>Material type</th>
<th>Principal properties</th>
<th>Common uses</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS</td>
<td>Good impact, strength, rigidity, heat stability, high</td>
<td>Housings for appliances, electronic and computer</td>
</tr>
</tbody>
</table>
gloss and resistance to aging. equipment. Automotive grills instrument clusters and body panels.

Properties Chart for Injection Molding Grade Plastics

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Molecular Type</th>
<th>Solid Density (g/cm³)</th>
<th>Melt Density (g/cm³)</th>
<th>Shrinkage (%)</th>
<th>Drying Requires (hours @ °C)</th>
<th>Melt Temp. (°C)</th>
<th>Mold Temp. (°C)</th>
<th>No flow temp. (°C)</th>
<th>Typical Ink. Press. (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS</td>
<td>Amorphous</td>
<td>1 - 1.2</td>
<td>0.9</td>
<td>0.5 - 0.6</td>
<td>2-3 @ 88-77</td>
<td>195 - 240</td>
<td>38 - 93</td>
<td>135 - 150</td>
<td>120 - 140</td>
</tr>
</tbody>
</table>

ABS E-module

<table>
<thead>
<tr>
<th>Material</th>
<th>Tensile Modulus (Young's Modulus, Modulus of Elasticity)</th>
<th>Ultimate Tensile Strength</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABS plastics</td>
<td>1.4-3.1 (10⁶ psi, 10⁹ N/m², GPa)</td>
<td>40 (10⁶ N/m², MPa)</td>
</tr>
</tbody>
</table>

Reasons to work with ABS:

- ABS's light weight and ability to be injection molded and extruded make it useful in manufacturing products.
- The most important mechanical properties of ABS are impact resistance and toughness. A variety of modifications can be made to improve impact resistance, toughness, and heat resistance.
Shot and part weight

- **Shot weight Definition**

  The shot weight is defined as the mass of plastic delivered in one complete filling of the mold, including the molded parts, sprue, runners, and flash.

**Shot weight equation**

<table>
<thead>
<tr>
<th>Shot Weight</th>
<th>=</th>
<th>Filling System</th>
<th>+</th>
<th>Part (\times n)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Runners</td>
<td>+</td>
<td>Gates</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>+</td>
<td>Sprue</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>+</td>
<td>Cold Well</td>
</tr>
</tbody>
</table>

Shot weight = \((g/cm^3) + (g \times 1)\) 

\[= g = \text{oz}\]

Total Shot weight = [oz]

- **Calculating part weight**

  Mass = grams

  \[\text{Volume} = \text{mm}^3 = \text{cm}^3\]

  Density = \(g/cm^3\) (ABS)

  \[m = \text{cm}^3 \times \text{g/cm}^3 = \text{g} \times 1\text{oz} / \text{g} \cong \text{oz} \cong \text{oz}\]

  \[\Rightarrow \text{Total part weight (single): oz (~g)}\]
Material required and Cavity number

- Material: Acrylonitrile butadiene styrene (ABS)
- 2880 Cycles per day
- 117.304g per cycle

\[
\text{Qty} = 2,880 \text{ cycles} \times 117.304 \text{ g/cycle} \approx 337.835 \text{ kg} \approx 350 \text{ kg}
\]

- Acrylonitrile butadiene styrene Cost

\[
\$31 \text{ USD/kg} \Rightarrow 29.795 \text{ TWD/kg} \times 350 \text{ kg} = 10,428.25 \text{TWD} \approx 10,500 \text{TWD}
\]

- Core and Cavity Size

\[
V = (476 \times 264 \times 69.94) \text{ mm} = 8,788,940.16 \text{ mm}^3 = 8,788.94 \text{ cm}^3
\]

\[S420J2 \approx 380 \text{TWD/kg}\]

\[\rho_{\text{steel}} = 7.85 \text{ g/cm}^3\]

- Core and Cavity Cost

\[
\rho_{\text{steel}} \times V_{\text{workpiece}} \times 1\text{kg}/1000\text{g} \times S55C \text{TWD/kg}
\]
Cost equation =

ABS material

Core and cavity plates

\[
\text{Cost} = 7.85 \, \text{g/cm}^3 \times 8,788.94 \, \text{cm}^3 \times 1\, \text{kg/1000g} \times 800 \, \text{TWD/kg} = 55,194.54 \, \text{TWD} \approx 55,200 \, \text{TWD}
\]

Injection machine selection

\[W = 4.3\, \text{oz} \times 1.20 = \frac{5.2}{8.0} \, \text{oz}\]

- Table 2

**Injection machine specifications**

<table>
<thead>
<tr>
<th></th>
<th>JM12MKIII-C</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Injection Unit</strong></td>
<td></td>
</tr>
<tr>
<td>Shot weight (PS)</td>
<td>oz 16 12</td>
</tr>
<tr>
<td></td>
<td>gm 454 341</td>
</tr>
<tr>
<td><strong>Injection Pressure</strong></td>
<td>kgf/cm² 1130 1504</td>
</tr>
<tr>
<td><strong>Screw Stroke</strong></td>
<td>mm 180</td>
</tr>
<tr>
<td><strong>Screw Speed Range</strong></td>
<td>RPM 164</td>
</tr>
<tr>
<td><strong>Plasticizing Capacity</strong></td>
<td>kg/hr 163 109</td>
</tr>
<tr>
<td><strong>Hopper Capacity</strong></td>
<td>litre 45</td>
</tr>
<tr>
<td><strong>Screw L/D Ratio</strong></td>
<td>mm/mm 16 18.6</td>
</tr>
<tr>
<td><strong>Injection Rate</strong></td>
<td>gm/sec 171.7 129</td>
</tr>
<tr>
<td><strong>Clamping Unit</strong></td>
<td></td>
</tr>
<tr>
<td>---------------------------</td>
<td>----------</td>
</tr>
<tr>
<td>Mould Clamping Force</td>
<td>Mp</td>
</tr>
<tr>
<td>Opening Stroke</td>
<td>mm</td>
</tr>
<tr>
<td>Space Between Tie Bar (H x V)</td>
<td>mm</td>
</tr>
<tr>
<td>Mould Thickness (Min-Max)</td>
<td>mm</td>
</tr>
<tr>
<td>Maximum Daylight</td>
<td>mm</td>
</tr>
<tr>
<td>Hydraulic Ejector Force</td>
<td>Mp</td>
</tr>
<tr>
<td>Hydraulic Ejector Stroke</td>
<td>mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Power/Heating Unit</strong></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Pump Motor</td>
<td>KW [Hp]</td>
<td>18.75[25]</td>
</tr>
<tr>
<td>Heating Capacity</td>
<td>KW</td>
<td>13.3</td>
</tr>
<tr>
<td>System Pressure</td>
<td>kgf/cm²</td>
<td>145</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Others</strong></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Dry Cycle Time</td>
<td>sec</td>
<td>2.8</td>
</tr>
<tr>
<td>Floor Space (L x W x H)</td>
<td>m x m x m</td>
<td>3.9 x 1.2 x 1.7</td>
</tr>
<tr>
<td>Oil Tank Capacity</td>
<td>litre</td>
<td>150</td>
</tr>
<tr>
<td>Machine Weight (Dry)</td>
<td>ton</td>
<td>3.4</td>
</tr>
</tbody>
</table>
Figures 16. Eject pins in the cavity (a and b)
Figures 17. Complete mold in wireframe (a and b)
Figures 18. Wireframe and solid section views of the complete mold (a,b,c)
Figures 19. Section views of the mold with the plastic part (a, b, c)
8. PowerMILL Simulations for the Cavity and Core

Rough Machining [1]
Finishing [1]
9. Conclusion

Plastics are commonly used for many products, electronics, in cars, food utensils, toys etc. The plastic materials are made from polymers and are produced in the temperature at which they become plastic in order to obtain the finished product. The most common technology of plastic parts is injection.

The main factors or disadvantages that affect injection molding were listed as follow: High initial tooling cost, part design restrictions and accurate cost. The main process factors are as follow: temperature, injection pressure, injection time, cooling time, melting temperature, mold temperature, filling time etc.

Accordingly from the previous information the results presented refer to: The advantages of injection molding are fast production, material and color flexibility, design flexibility and low waste. To make a very good and accurate mold is necessary it has all basic elements to work properly: sprue, gates, runners and vents, ejection pins, cooling system.

Other part of the paper refers in detail about the injected part, properties of injection molding, drawings of the mold and simulation of the core and cavity using PowerMILL program.
10. Glossary of Some Plastic Injection Molding Terms

ANSI: Abbreviation for American National Standards Institute

Aspect Ratio: Ratio of total flow length to average wall thickness.

Back Pressure: The resistance of the molten plastic material to forward flow. In molding, back pressure increases the temperature of the melt, and contributes to better mixing of colors and homogeneity of the material. However, as back pressure increases, so does cycle time.

Backflow: molten resin flows back out of the mold, returning to the runners.

Backing Plate: A plate used as a support for the mold cavity block, guide pins, bushings, etc.

Balanced Runner: A runner system designed to place all cavities at the same distance from the sprue.

Blow Molding: Method of fabrication in which a warm plastic hollow tube is placed between the two halves of a mold cavity and forced to assume the shape of that mold cavity by use of internal pressure. This process forms hollow articles such as bottles, tanks, etc.

Boss: A raised feature of a molded part designed to add strength, facilitate alignment during assembly or for attachment to another part.

Brighteners: Are used to add smoother or brighter coatings or finishes.

Burned: Showing evidence of excessive heating during processing or use of a plastic, as evidenced by blistering, discoloration, distortion or destruction of the surface.

Casting: The process of forming solid or hollow articles from fluid plastic mixtures or resins by pouring or injecting the fluid into a mold or against a substrate with little or no pressure, followed by solidification and removal of the formed object.

Cavity: A depression, or a set of matching depressions, in a plastics-forming mold which forms the outer surfaces of the molded articles.

Clamp: The part of an injection molding machine incorporating the platens that provides the force necessary to hold the mold closed during injection of the molten resin and open the mold to eject the molded part.
Clamping Area: The largest rated molding area an injection press can hold closed under full molding pressure.

Clamping Force: The force applied to the mold to keep it closed, in opposition to the fluid pressure of the compressed molding material within the mold cavity and the runner system.

Clamping Plate: A plate fitted to a mold and used to fasten the mold to a platen. Clamping Pressure: The pressure applied to the mold to keep it closed during the molding cycle.

Clarifiers: Additive used in resins to improve transparency or translucency.

Closed-loop Control: System for monitoring and automatically adjusting injection molding process conditions such as temperature, pressure and time. The automatic changes keep part production within preset tolerances.

Coefficient of Thermal Expansion (CTE): The change in length of a material for a unit change in temperature, per unit of length.

Co-Injection: Simultaneous or near simultaneous injection of multiple materials.

Cold Flow Lines: Imperfections within the part wall due to thickening or solidification of resin prior to full cavity fill.

Cold Molding: The process of compression molding involving shaping an unheated compound in a mold under pressure then heating the article to cure it.

Cooling Channels: Channels located within the body of a mold through which a cooling medium is circulated to control the mold surface temperature.

Cycle Time: The time required by an injection molding system to mold a part and return to its original position/state.

Degradation: A deleterious change in the chemical structure, physical properties or appearance of a plastic caused by exposure to heat, light, oxygen, weathering or other external influence.

Elasticity: The ability of a material to quickly recover its original dimensions after removal of a load that has caused deformation.
Extrusion: The process of forming continuous shapes by forcing a molten plastic material through a die.

Injection Blow Molding: Blow molding process by which the plastic parison to be blown is formed by injection molding.

Injection Molding Pressure: The pressure applied to the cross-sectional area of the molding cylinder.

Injection Molding: The method of forming objects from granular or powdered plastics, most often of the thermoplastic type, in which the materials is fed from a hopper to a heated chamber in which it is softened, after which a ram or screw forces the material into a mold. Pressure is maintained until the mass has hardened sufficiently for removal from the mold.

Injection Pressure: The pressure on the face of the injection screw or ram when injecting material into the mold, usually expressed in PSI.

Moving Platen: The platen of an injection molding machine that is moved by a hydraulic ram or mechanical toggle.

Multi-Cavity Mold: A mold having two or more impressions for forming finished items in one machine cycle. Multidirectional flow: flow direction changes during filling resulting in orientation in different directions which can cause flow marks, stresses and warping. Multiple cavity mold: produces more than one identical part with each cycle. Multi-Shot Molding: The injection of two-or-three materials, in sequence, into a single mold during a single molding cycle. The injection molding machine is equipped with two-or-three plasticators. (See also co-injection)
11. References


[4] https://www.lubrizol.com/.../Processing-Guides/Injection-Molding...

