



Industrial Guideline for Injection Moulding : Case of Jaundice Meter

Naqibah Azman¹, Nik Hisyamudin Muhd Nor¹, Mohd Fahrul Hassan¹, Mohd Hafiz Bin Burhan¹, Siti Zarina Muhd Muji², Fariza Mohamad², Faiz Mohd Turan³, Abdul Rahman Bahasa⁴, Yokoyama Seiji⁵

¹Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Malaysia, naqibahazman@gmail.com, nhisyam@uthm.edu.my, fahrul@uthm.edu.my, hafizb@uthm.edu.my

²Faculty of Electtical and Electronic Engineering, Universiti Tun Hussein Onn Malaysia, Malaysia, szarina@uthm.edu.my, farizamd@uthm.edu.my

³Faculty of Manufacturing Engineering, Universiti Malaysia Pahang, Malaysia, faizmt@ump.edu.my

⁴Recove Group Sdn. Bhd., Malaysia, irrahmanb@recovegroup.com

⁵Faculty of Mechanical Engineering, Toyohashi University of Technology, Hibari-Gaoka, Tenpaku-cho, Toyohashi-shi, Japan, yokoyama@me.tut.ac.jp

ABSTRACT

In designing industries, a lot of measures need to be taken before coming to the final design phase. During the initial stage of designing a product, designers usually will design according to their sketching without considering guidelines for mass production manufacturing. This will consume so much time on re-designing for to suit manufacturing requirement. There-fore this paper purpose to consider industrial guideline specifically on designing product directly for injection moulding. The most basic requirement on design for direct manufacturing that designers need to consider while designing are wall thick-ness, rib, boss and cored holes, suggested design for corners and materials selection. The guideline was applied to a jaun-dice meter prototype design as a case study.

Key words : Jaundice meter, injection moulding, engineering design.

1. INTRODUCTION

The previous century has watched the rapid increment of plastics and their development into all business sectors. As per world utilization of crude materials by weight, plastic is the most significant in analysis with other old materials, for example, aluminum, steel, elastic, copper, and zinc, because of its properties and lower manufacture cost [1], [2].

Injection moulding is amongst the most critical procedures for manufacturing of plastic items and roughly 33% of all plastics are changed over into parts utilizing infusion forming forms (Tang, Kong, Sapuan, Samin, and Sulaiman, 2006). The use of infusion forming forms is expanding fundamentally in numerous enterprises like bundling, aviation and flying, building and construction, car parts and family unit articles [1], [3], [4].

The plan and fabricate of infusion formed polymeric parts with wanted properties is an expensive procedure ruled by empiricism, including the rehashed adjustment of genuine tooling. Among the errand of shape configuration, planning the form particular strengthening geometry, generally on the center side, is very convoluted by the inclusion of projection and melancholy [3].

The final quality of injected parts in injection moulding depends on the material characteristics, the mold design and the process conditions [5]. There are three fundamental operations in injection moulding. First, the plastic granules will be converted into a melt. Then, the molten plastic will be injected into the mold cavity or cavities under pressure via sprue, runner and gate systems and finally, the mold tools will be opened to eject the part from the cavities [1]

A successful application of an engineering thermoplastic requires more than identifying a specific product or grade. Three areas – design, product, process – are all interrelated and the appropriate rules in each area must be followed to ensure a successful application. In most cases, the process must be determined before a specific resin grade can be selected. During this review, designers also need to consider whether the process is capable of meeting the design requirements such as size, shape, detail and tolerance [6].

In order to design a mold, many important designing factors must be taken into consideration. These factors are mold size, number of cavity, cavity layouts, runner systems, gating systems, shrinkage and ejection system [3]

In this paper the design guidelines for plastic injection moulding to minimize modification time and cost effective as a whole is presented.

2. DESIGN GUIDELINES

This section illustrates the design guidelines for plastic injection moulding.

2.1. Wall Thickness

According to [7], maintain a wall thickness of less than 5mm because thick walls can lead to long cycle times and poor mechanical properties. Avoid large variations in wall thicknesses in order to simplify flow pattern and minimize variations in shrinkage that can lead to warpage. Avoid abrupt changes in wall thickness, as this can create stress concentration areas that may reduce a part’s impact strength. Wall thickness changes should have transition zones that reduce the possibility of stress concentrations, sinks, voids, and warp. Draft angle for the wall thickness 3 degree is a recommended wall thickness.

2.2. Ribs

Maximum rib thickness should be 50%-75% of the nominal wall because if the rib is too thick, it will cause sink marks. To avoid thin sections of steel in your core or cavity which may cause it to break, the distance between ribs should be at least two and a half times the nominal wall thickness [6]. Rib thickness should be between 0.5 to 0.6 times nominal wall thickness to avoid sink mark. Rib height should be 2.5 to 3 times nominal wall thickness.

Draft angles : 0.5-1.5 degree

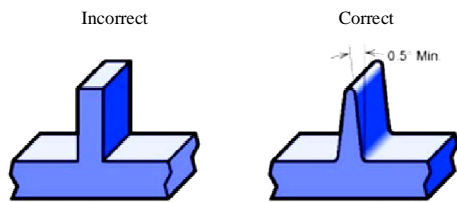


Figure 1 : Draft angle guidelines [8]

Figure 1 shows, rib should have 0.5 to 1.5-degree draft angle to facilitate ejection. Rib base should have radius 0.25 to 0.4 times nominal wall thickness. Distance between two ribs should be 2 to 3 times (or more) nominal wall thickness.

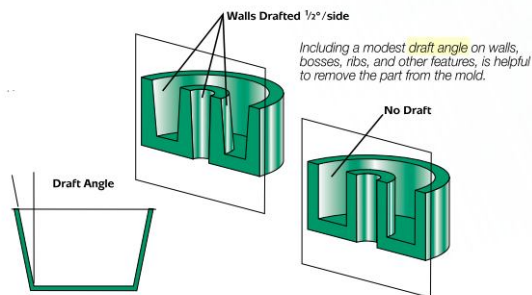


Figure 2 : Cross section figure applying draft on walls

Figure 2 shows draft applied on the walls helps the part come out of mold more easily. If no draft is applied, the part will either stuck or cannot be remove at all from the mold.

2.3. Boss and cored holes

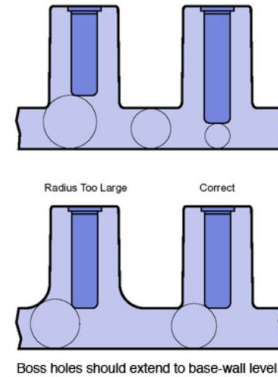


Figure 3 : Depth of cored holes [1]

Bosses provide cored holes that can be used for mechanical fastener locations. The boss distributes bearing loads and transfers fastener loads to the main structure. Excessively, high bosses tend to trap air, so boss height normally should not exceed twice the boss diameter. Slenderness ratio of a cored hole is its depth divided by diameter as shown in Figure 3. [1]

Normally, the boss hole should extend to the base-wall level, even if the full depth is not needed for assembly. Shallower holes can leave thick sections, resulting in sink. Deeper holes reduce the base wall thickness, filling problems, knit lines, or surface blemishes. The location and quantity of ribs is vital in avoiding exacerbating problems the ribs were intended to correct— e.g., ribs added to increase part strength and prevent breakage may reduce the ability of the part to absorb impacts without failure. Furthermore, a grid of ribs added to ensure part flatness may lead to mold-cooling difficulties and warpage.

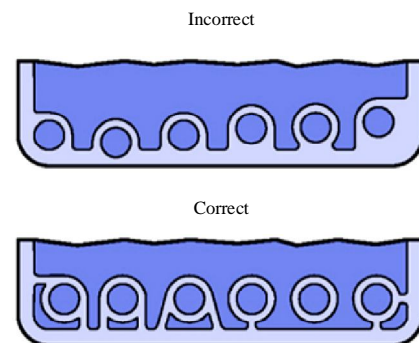


Figure 4 : Guide on rib for bosses cored holes

Shown in figure 4, the location of bosses cored holes should not be too close to the wall. This may cause sinkmarks on the outside because the wall will be thicker than recommended. The rib should not be too thin because it will be more breakable.

2.4 Design for Corners

Guidelines for corners design according to the above figure, the inside radius should be set constant as the wall thickness. If the radius centerpoint for inner and outer radius are not concentric with each other, this will cause the corners to be too thin or too thick. As for thin corners, this will cause it to be tenuous while for thick corners will cause increase the cost of material used in the product.

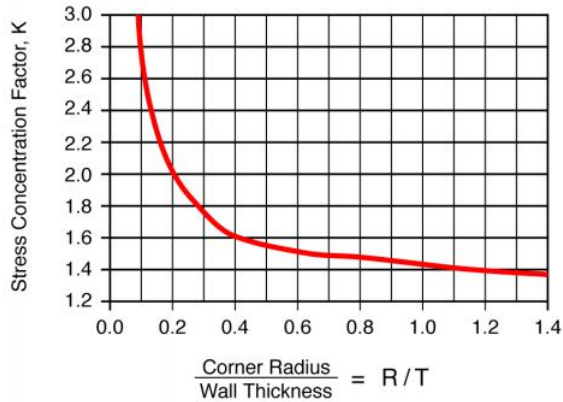


Figure 5 : Stress Concentration Factor, K [6]

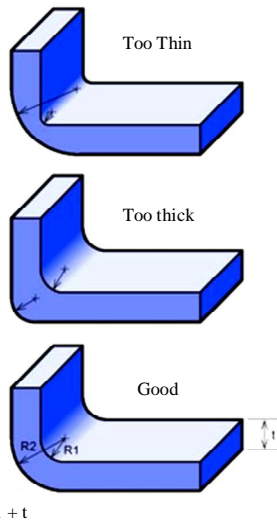


Figure 6 : Guide for designing corners

Sharp corners greatly increase stress concentration, which, when high enough, can lead to part failure. In figure 5 shows as the radius increase, the stress decrease. Therefore, it is recommended that an inside radius be a minimum of one times the thickness.[6] Figure 6 shows guidelines for designing corners where thin corners will cause the corners to be easily breakable but when the corner is too thick it will be waste of material.

3. MATERIAL GUIDELINES

Resin	Recommended Wall Thickness (Inches)	Recommended Wall Thickness (mm)
ABS	0.045 – 0.140	1.143 - 3.556
Acetal	0.030 – 0.120	0.762 - 3.048
Acrylic	0.025 – 0.150	0.635 - 3.81
Liquid crystal polymer	0.030 – 0.120	0.762 - 3.048
Long-fiber reinforced plastics	0.075 – 1.000	1.905 - 25.4
Nylon	0.030 – 0.115	0.762 - 2.921
Polycarbonate	0.040 – 0.150	1.016 - 3.81
Polyester	0.025 – 0.125	0.635 - 3.175
Polyethylene	0.030 – 0.200	0.762 - 5.08
Polyphenylene sulfide	0.020 – 0.180	0.508 - 4.572
Polypropylene	0.025 – 0.150	0.635 - 3.81
Polystyrene	0.035 – 0.150	0.889 - 3.81
General rule of thumb	0.040-0.140	1.016 - 3.556

Figure 7 : Common material used in injection moulding[9]

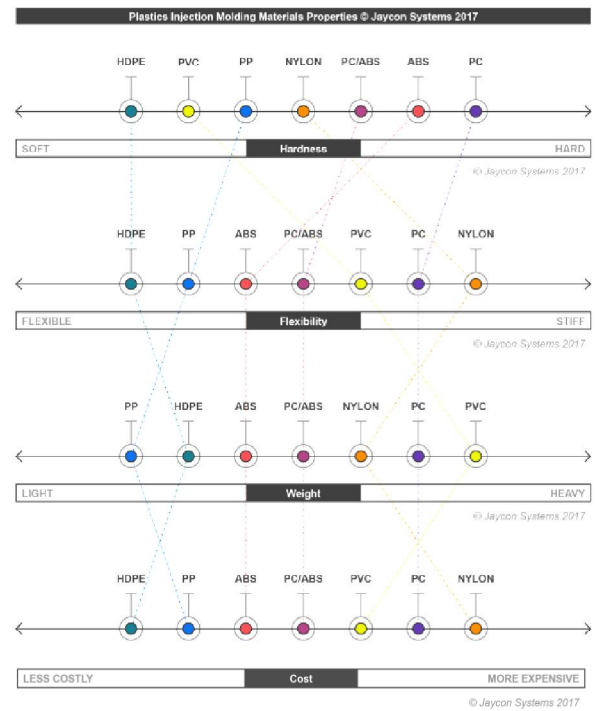


Figure 8 : Material use in Injection Moulding[10]

In the above figure 7 shows the common material used in injection moulding and its properties. It shows recommended thickness for each material according to their material properties. Among the most commonly used material for injection moulding is Acrylonitrile Butadiene Styrene (ABS) [11]. ABS is a thermoplastic material known for its high impact and toughness. This is shown in figure 8 that ABS are high in strength (hardness), low flexibility, light weight and inexpensive.

4. CASE STUDY: JAUNDICE METER

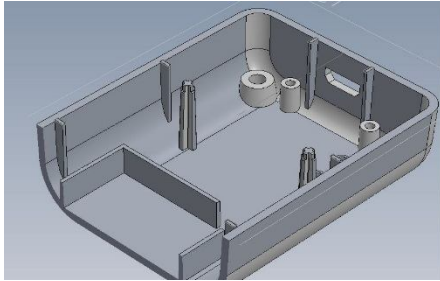


Figure 9 : Cross-section of jaundice meter

Figure 9 shows jaundice meter have gone through changes in design from preliminary design to design which follows injection moulding design specification set by German Malaysia Medical Industry (GMMI) bound to ISO 13485. Designers usually draw design for 3D printing to be used as prototype without considering any design guidelines on injection moulding. The material use for this prototype is ABS

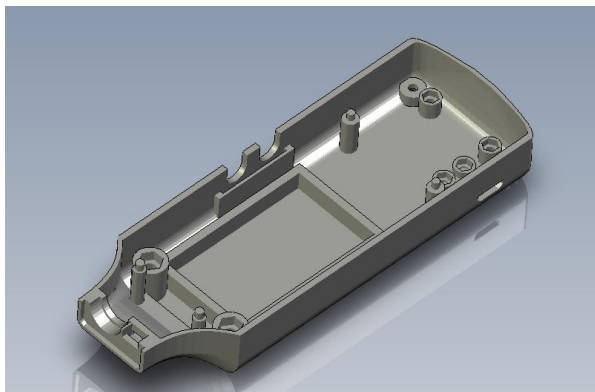
1st Generation of Jaundice Meter



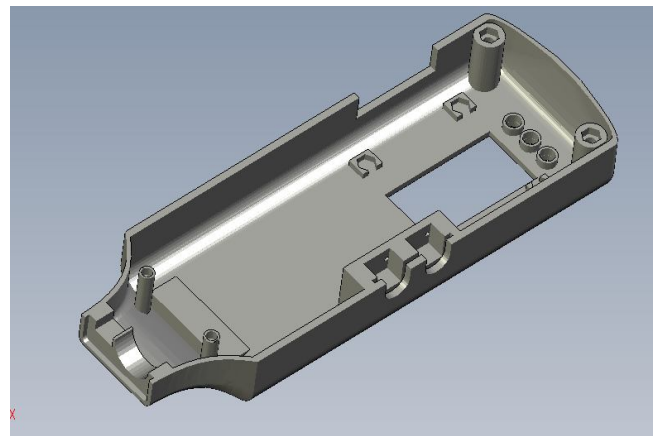
Figure 10 : Non-invasive jaundice meter (Gen 1)

Figure 10 shows the initial lab scale design of the jaundice meter which have been used by previous researcher and handmade using wood as a case.

2nd Generation Jaundice Meter



(a) Bottom Housing



(b) Top Housing

Figure 11 : (a) Bottom housing (b) Top housing of a jaundice meter primarily

Figure 11 shows the design of jaundice meter formerly which does not follow injection moulding guidelines. Designers from research sector does include injection moulding constrain during design process causes more time and money to alter the design. This is because most of researchers build prototype project by using 3D printing thus accurate dimension will produce without have to consider tolerance or draft angles. This paper purposely wrote to advise designers to always consider these guidelines if the product design is going for mass production.

3rd Generation Jaundice meter

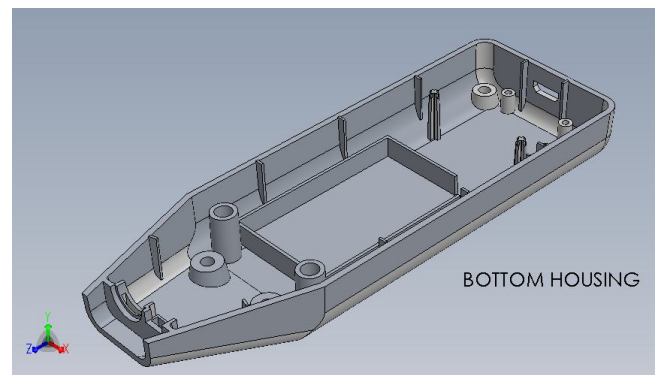


Figure 12 : Bottom housing jaundice meter (Standardize)

Figure 12 shows the re-design jaundice meter according to the industrial guidelines. The usage of material decrease thus decrease the material cost. It is made to be more functional and reliable as a whole. Ribs are added around the internal case to ensure that it is tight when close. Ribs around the internal wall also help the casing from misalignment during drops or strong impact exerted.

5. SUGGESTION

While designing plastic part, pitfalls in achieving quality, consistency and productivity must be considered. It is wrong to assume that shapes can be molded successfully without any

defects. All shapes may not be 100% moldable. To improve the foldability injection moulding process has to be understood in depth.

Part design obviously has to be influenced by the intricacies of the process. Filling phase of the process is influenced by type of gate, location of gate, number of gates, size of gate (also dependent on material viscosity). Gate should be located at such a position from where flow path to thickness ratio (flow ratio) is constant in all direction. The difference in flow ratio could be as small as possible. In some cases where thickness variation is unavoidable, melt must flow from thin section to thick section for better mouldability. Melt flow from thin to thick results in poor moulding. It should be realized that variation in wall thickness, hole / slot, variation of mold surface temperature introduces variation in resistance to flow. Therefore melt moves in number of streams with different velocity in different direction and mold does not fill in balanced manner.

6. CONCLUSION

The aim of this work is to promote industrial guidelines to mechanical designers so that they can save time in redesigning it for injection moulding process. The case study used in this paper is set as an example in following industrial guidelines.

ACKNOWLEDGEMENT

We would like to express our gratitude to Research Management Centre (RMC), Universiti Tun Hussein Onn Malaysia for the funding provided to this research work under vot U803 GPPS.

REFERENCES

- [1] Huamin Zhou, *Computer Modeling For Injection Moulding :Simulation, Optimization, and Control*. 2013. <https://doi.org/10.1002/9781118444887>
- [2] A. Salimi, M. Subaşı, L. Buldu, and Ç. Karataş, “Prediction of flow length in injection moulding for engineering plastics by fuzzy logic under different processing conditions,” *Iran. Polym. J. (English Ed.)*, vol. 22, no. 1, pp. 33–41, 2013. <https://doi.org/10.1007/s13726-012-0103-5>
- [3] S. H. Tang, Y. M. Kong, S. M. Sapuan, R. Samin, and S. Sulaiman, “Design and thermal analysis of plastic injection mould,” *J. Mater. Process. Technol.*, vol. 171, no. 2, pp. 259–267, 2006.
- [4] M. Altan, “Reducing shrinkage in injection mouldings via the Taguchi, ANOVA and neural network methods,” *Mater. Des.*, vol. 31, no. 1, pp. 599–604, 2010.
- [5] N. Khoshooee and P. D. Coates, “Application of the Taguchi method for consistent polymer melt production in injection moulding,” *Proc. Inst. Mech. Eng. Part B J. Eng. Manuf.*, vol. 212, no. 8, pp. 611–620, 1998. <https://doi.org/10.1243/0954405981515888>
- [6] I. Stratasys Direct, “Injection Moulding Design Guidelines,” *Strat. Direct, Inc.*, pp. 1–8, 2015.
- [7] C. RTP Company, “Part Design Guidelines for Injection

- Molded Thermoplastics,” pp. 1–5, 2017.
- [8] ICOMold, “Wall Thickness by Resin Material Guidelines,” pp. 1–10, 2018.
- [9] Prabodh C. Bolur, *A Guide To Injection Moulding Of Plastic*. 2000.
- [10] K. Ramesh, A; Sivaramanarayanan, “An Overview of the Plastic Material Selection Process for Medical Devices,” no. February, pp. 1–25, 2013.
- [11] J. B. Tranter, P. Refalo, and A. Rochman, “Towards sustainable injection moulding of ABS plastic products,” *J. Manuf. Process.*, vol. 29, pp. 399–406, 2017. <https://doi.org/10.1016/j.jmapro.2017.08.015>