Jurnal
Teknologi

Full Paper

PRODUCT AND TOOLING DESIGN OF SLANTED GLASS INJECTION MOULD FOR VISUALIZATION OF FLOW MOLTEN PLASTIC

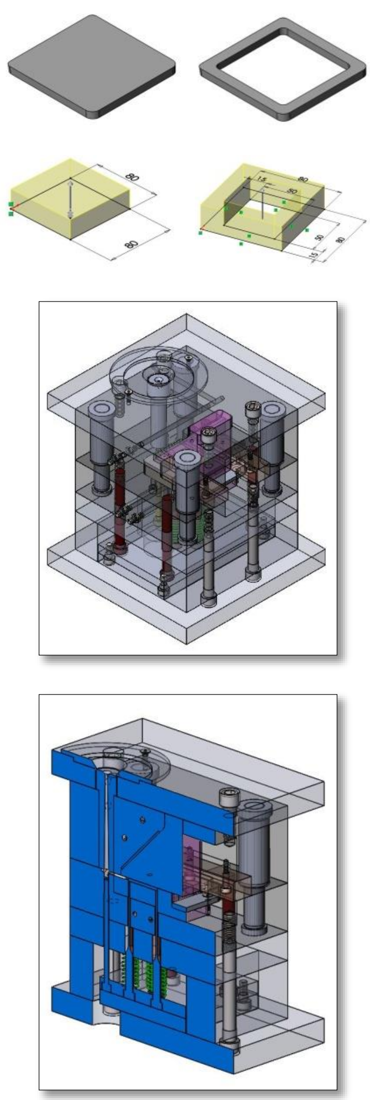
Article history
Received
15 May 2015
Received in revised form
12 September 2015
Accepted
30 September 2015

M. Amran*, M. Faizal, M. Shahir, R. Izamshah, Sivaraos

Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

*Corresponding author
mohdamran@utem.edu.my

Graphical abstract



Abstract

A two-plate plastic injection mould having a new method of embedded slanted glass at mould cavity side is designed where it can be used to visualize the flow of molten plastic material moving inside the mould cavity. Due to observation of molten plastic materials cannot be done in actual mould made from metal, therefore slanted glass mould is developed. This project shows the design process of slanted glass mould from initial plastic product design until mould tooling design. The plastic product and the slanted glass mould were designed using SolidWorks software. In designing and producing the injection molded plastic product, various factors are needed to be considering such as corner radius, draft angle and the product wall thickness Two different geometrical shapes of plastic products have been designed that were the flat and ring plastic product. Meanwhile for mould tooling design, the method to capture the flow pattern of molten plastic materials is the most considering factor in this project. Others considering factors included are location of feeding system, tolerance fitting and water cooling system. Thus, this project helps mould designer the important of design process should be considered during plastic design and tooling design especially for slanted glass injection mould.

Keywords: Plastic product; injection moulding; slanted glass mould; flow visualization

Abstrak

Dua plat acuan suntikan plastik yang mempunyai kaedah baru kaca condong tertanam di acuan sebelah rongga direka di mana ia boleh digunakan untuk menggambarkan aliran bahan plastik cair bergerak di dalam rongga acuan. Oleh kerana pemerhatian bahan plastik cair tidak boleh dilakukan dalam acuan sebenar yang dibuat dari logam, oleh itu acuan kaca condong dibangunkan. Projek ini menunjukkan proses reka bentuk acuan kaca condong dari awal reka bentuk produk plastik sehingga reka bentuk perkakasan acuan. Produk plastik dan acuan kaca condong direka menggunakan perisian Solidwork. Dalam mereka bentuk dan menghasilkan produk suntikan acuan plastik, pelbagai faktor perlu dipertimbangkan seperti jejari di bahagian sudut produk, draf sudut dan ketebalan dinding produk plastik. Dua bentuk geometri yang berbeza untuk produk plastik telah direka iaitu produk rata dan cincin. Sementara itu, bagi reka bentuk perkakasan acuan, kaedah untuk menangkap corak aliran bahan plastik lebur adalah faktor yang paling diutamakan dalam projek ini. Selain dari itu, pertimbangan faktor lain adalah lokasi sistem memberi makan, toleransi sistem pemasangan dan penyejukan air juga diambil kira. Oleh itu, projek ini dapat membantu pereka acuan mengenai perkara yang penting dalam proses reka bentuk yang perlu dipertimbangkan semasa reka bentuk plastik dan reka bentuk alat terutamanya untuk kaca condong suntikan acuan.

Kata kunci: Produk plastik; pengacuan suntikan; acuan kaca condong; aliran visualisasi

1.0 INTRODUCTION

It is important to understand the polymer flow phenomena in molten stage during mould cavity filling. This is because of economical reason such as reducing the plastic part defects as well as increasing the quality requirements of moulded parts. Therefore, it is useful to carry out research both in visualization [1] and simulation [2] during flow of molten polymer during injection moulding process [3]. Recently, the polymer flow using visualization techniques in plastic injection mould has been studied by several researchers. Bociga and Jaruga investigated the polymer melt in the cavity filling with various accesses and found that visualization technique had been able to improve various problems that cannot be seen during simulation [4]. Kanetoh and Yokoi studied the flow fronts of plastics melt using glass insert mould. They used the direct visualization method and found that defect of flow marks was observed at plastic parts where it was closely related to plastic behaviour in the flow front [5]. Result from simulation can be compared with visualization where the parameters of machine injection can be optimized using various optimization method such as using full factorial [6], response surface method [7] and Taguchi method [8].

In this project, two plastic parts has been selected for design. Selections of two types of these plastic parts are based on the thickness, shape, and its suitability in producing a plastic injection mold for use in the future research. In future research, plastic parts that are designed in this project, are taken into consideration the plastic flow that can cause product defects such as weld lines, meld lines, air trapped and geometrical size of the product. The product shape will be the cavity shape for the slanted glass mould. Furthermore, this project discusses the design mechanism of slanted glass mould that will be used for visualize the molten polymer flow through transparent glass. There are three locations of digital camera to view of the molten polymer, firstly at the surface front view, secondly at surface side view and lastly at the surface top view of the plastic product.

1.1 Product Design

Good design is important for any manufactured product but for plastics it is absolutely vital. There are so many considerations to be look of in order to get good quality finish product. However, according to the two main plastic parts that introduced, the consideration about the good and bad corner design, the stress concentration factor and also the wall thickness are to be look.

1.1.1 Corner Design

Walls usually meet at right angles, at the corners of a box for example. Where the box walls meet the base, the angle will generally be slightly more than 90° because of a draft angle on the walls. However, this can causes two difficulties. The first difficulty is that the increase in thickness at the corner breaks the rule of uniform wall thickness. The maximum thickness at a sharp corner is about 1.4 times than the nominal wall thickness as shown in Figure 1. The result is a longer cooling time accompanied by a risk of sink marks and warping due to differential shrinkage.

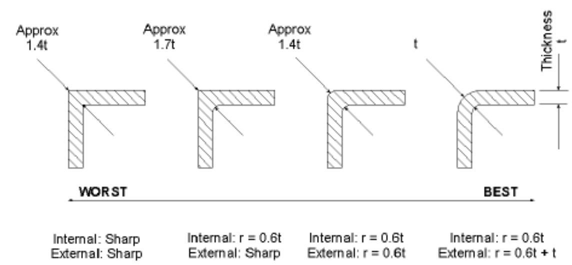


Figure 1 Good and bad corner design [2]

The other difficulty is even more serious. Sharp corners concentrate stress and greatly increase the risk of the part failing in service.

Plastics are said to be notch-sensitive because of their marked tendency to break at sharp corners. This happens because the stress concentration at the corner is sufficient to initiate a microscopic crack which spreads right through the wall to cause total failure of the part. Sharp internal corners and notches are the single most common cause of mechanical failure in moulded parts [9]. There is some solution for this problems, first is to avoid sharp internal corners. Second is to make sure that internal radii should be at least 0.5 and preferably 0.6 to 0.75 times the wall thickness. On the other hand is to keep corner wall thickness as close as possible to the nominal wall thickness. Ideally, external radii should be equal to the internal radii plus the wall thickness [10].

By properly designed corners will make a big difference to the quality, strength and dimensional accuracy of a moulding parts such as smooth curved corners will help plastic flow in the mould by reducing pressure drops in the cavity and minimizing flow-front break-up.

1.1.2 Ribs

When the normal wall thickness is not stiff enough or strong enough to stand up to service conditions, the part should be strengthened by adding ribs rather than making the whole wall thicker. The principle is the same with the 'I' and 'T' beam where they are almost as rigid as solid beams but are only a fraction of the weight and cost.

Ribs are used to improve the rigidity of a plastics part without increasing the wall thickness so much that it becomes unsuitable for injection moulding. Ribs are important in the design of plastics parts because they allow us to make a component rigid without making it too thick. The usual rule is to make the gap at least twice the nominal wall thickness and preferably three times or more [8]. The consideration is that, these ribs must not be too close together. This is because the gap between the ribs is produced by an upstanding core in the mould. If this core is too thin it becomes very difficult to cool and there may also be a shrinkage and warpage effect that will cause ejection problems. Figure 2 shows an example of ribs designed by using Solidworks software.

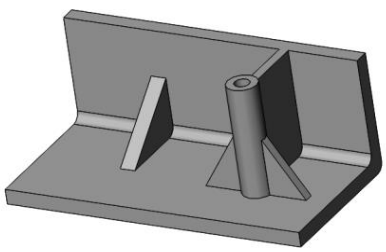


Figure 2 Example of ribs at plastic product [10]

1.1.3 Wall Thickness

Moulded plastics do not lend themselves to solid forms. The reason is, plastics are processed with heat but they are poor conductors of heat. This means that thick sections take a very long time to cool and so are costly to make [9]. The problems posed by shrinkage are equally severe. During cooling, plastics undergo a volume reduction. In thick sections, this either causes the surface of the part to cave in to form an unsightly sink mark, or produces an internal void. Furthermore, plastics materials are expensive, it is only high-speed production methods and net-shape forming that makes moulding viable. Thick sections waste material and are simply uneconomic.

Solid shapes usually do the job well in wood or metal usually transformed to a 'shell' form in plastics. This is done by hollowing out or 'coring' thick parts so the thin walls part can be made however, in order to achieve a strong part by only thin wall impossible, it must be reinforce by ribs [6,10]. In practice, there must be some variation in thickness to accommodate and fulfill functions or aesthetics. It is very important to keep this variation to a minimum. Figure 3 shows the gradual transition between thick and thin wall sections by considering reducing the defects. A plastics part with thickness variations will experience differing rates of cooling and shrinkage. The result is likely to be a part that is warped and distorted, where close tolerances become impossible to hold on the other hand, variations in thickness are unavoidable [6,9].

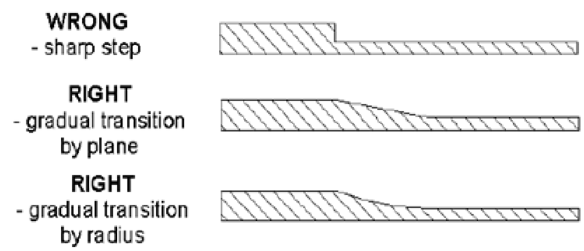


Figure 3 Gradual transitions between thick and thin sections [11]

1.2 Mould Design

No matter how good the design of the desired part with having consideration of many factor, if the mould to produce the part are not equivalent quality of course the product or part produce are not expected. The design of the slanted glass mould also considering many factors that is the sprue, runner and gate system, machining process, tolerance fitting, water cooling and the right assembly of the mould.

1.2.1 Sprue, Runner and Gate

The molten plastic transportation system begins with the injection moulding machine nozzle and continues into the mould, progressing through the sprue, runner gate, and the product forming cavity area. Each of these components has an impact on moulding system, which can be significant to quality of plastic product.

Start with the sprue; there are two main types of sprue and that are the hot sprue and the cold sprue. In fabricating the slanted glass mould, the direct cold sprue was used because this type of gate was commonly used for single cavity mould, where sprue feeds the molten plastic to mould cavity area directly and rapidly with minimum pressure drop [11].

The term runner refers to the molten plastic delivery channel between sprue and gate. The runner has the potential to influence the size, shape and the mechanical properties of the melded plastic parts [12]. There are five basic shapes of runner cross-section that are fully rounded, parabolic, trapezoid, half round and half trapezoid. The most efficient flow channel cross-section is the full round that resulting in the lowest pressure drop per material used. However the full round runner has the disadvantages of requiring the two halves of the runner to be machined in each of the two half of the mould. Furthermore, these two half runner must the closely match up to form the full round runner when the mould is closed. Due to the potential of miss alignment, the parabolic cross-section shape runner was selected in this project because it provides the closest performance to a full round runner without the need to machine and match up the two halves of a full round runner [11].

After the melt plastic material flowing through the sprue and runner, it must run through the gate before entering the cavity. Gate is used to allow for easy, potentially automatic separation of the part from the runner system while allowing for filling and packing at the same time. The shape, size and placement of the gate can significantly affect the ability to successfully mould a product. In fabricating the slanted glass mould, the gate that used was the common edge gate where it was the most basic type of gating system. It created in a rectangular shape in cross section and attached to the plastic product. This edge gate was selected because the gate will position directly centre with the edge of plastic product maintaining the moulded parts position and orientation on the runner. Further, this edge will provide the indication of weld line that can be observed for inspection the plastic product quality [13].

1.2.2 Machining Process

Plastic injection mould is the most difficult production tooling to produce which involves designing, machining and assembling process [13,14]. All the plates are needed to proceed to the squaring process by following the size needed in the list of procurement an also by referring to the technical drawing made by the design engineer. All the operation must follow exactly the technical drawing otherwise other problems may occur during the assembly stage. All the plate has been mark in order to indicate the reference to the mould that also called as a datum. This is important to ensure that every machining process must be taken from the datum [13,14] so that the assembly process can be done faster and without making any mistake. In this study, it is found that for fabricating the slanted glass injection mould the most difficult part to fabricate is the core plate because it needs to be very precise between the cooling channel and the ejector pins otherwise the cooling channel will leak. Meanwhile the easiest plate to fabricate is the spacer block.

1.2.3 Tolerance Fitting

Tolerance fitting is important in producing a precision plastic injection mould. Plastic injection mould itself needs a high precision part in order to product a high end product. If the dimension in the drawing was written without a tolerance, no one will know the importance or the unimportance of the mould part fabricated and leading to the undesirable result such as the improper fits that leads to delay which also leads to increasing the reworks cost. The tolerance and fitting that is used in fabricating the injection mould is following the ANSI B32.100-2005 standard. In the case of ejector pin, the standards are in the shaft basis where the code G7/h6 is followed because the conditions for the ejector pins are sliding. The tolerance between ejector pins and the plate are crucial, if the tolerance is too tight, the plate might grip the pins due to the thermal expansion and if the tolerance between them is too loose, plastic material might flow between the gaps and leading to the flashing defect. Inserts and plates also need the tighten tolerance in order to evade any error in assemble stage, defect in injection mould product processing and also might lead to the increasing of the rework cost and material waste. Figure 4 shows the shaft basis system and the hole basis system that is used to the insert with the code standard of H7/g6.

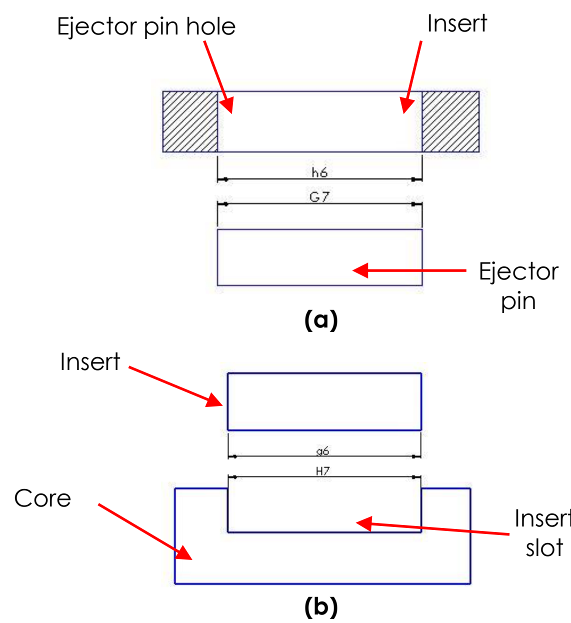


Figure 4 **(a)** Shaft basis system for G7/h6 standard
(b) Hole basis system for H7/g6 standard

1.2.4 Water Cooling

While operating the injection plastic mould, temperature is the main parameter that cannot be neglected. Temperature cannot completely eliminate however it can be reduced by having the water cooling channel. Injection moulding process is cyclic in characteristic. Cooling time is about 50 to 75% of the total cycle time [15]. Therefore, optimising cooling time for best performance is very important from quality and productivity point of view. Cooling channel design location, size and type should ensure that melt freezes uniformly inside the mould. The localization of the cooling channels has a great importance for having an uniform cooling, because it decides the moulding surfaces temperature distribution and evolution during the period of cooling. Cooling channel design can be perfected with the help of simulation analysis [12].

The injected material loses temperature in the contact with the mould surfaces, transferring itself heat through the mould. For speeding the heat transfer process, the mould designer design specific holes in the adjacent surfaces of the moulded part in the mould. These cooling channels constitute the cooling system of a mould.

Figure 5 shows the heat flow profile in mould where distance between the cooling channels and the moulding surface (h) and the distance between cooling channels (e) are the main parameters to be considered [16]. By referring to Figure 6(b), it shows that cooling channels are made through the ejector pin where the place was near to the plastic part.

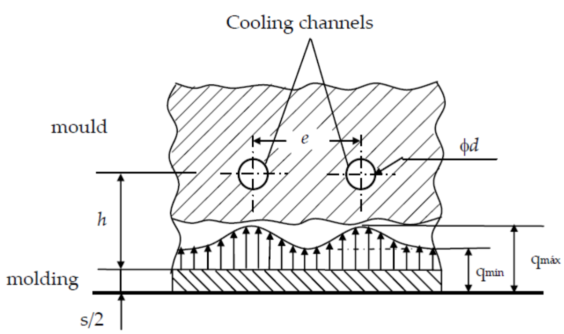


Figure 5 Heat flow profile

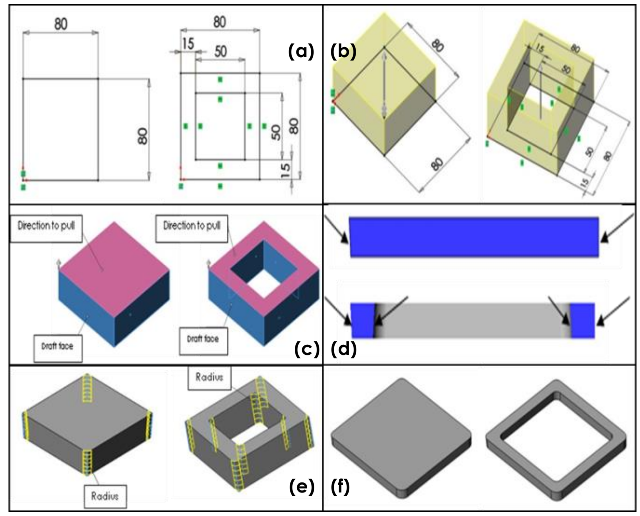


Figure 6 (a) Drafting **(b)** Extruding**(c)** Draft angle setting step **(d)** Cross section showing draft angle **(e)** filleting and **(f)** Finish part design for Flat and ring plastic part

2.0 PRODUCT AND INSERT MOULD DESIGN PROCESS

In designing the slanted glass mould, there are two design phases that is needed to be done. The first phase is designing the product or plastic parts by having consideration of several factors such as good and bad corner radius at plastic parts, the stress concentration factor and also the part wall thickness. The second phase is designing the slanted glass mould having the cavity area of the product designed.

2.1 Product design

In designing the product, there are some steps involving in order to achieve the finish 3D solid modeling. Figure 6 shows the five basic steps in order to design the plastic part. In this process, Solidworks software version 2013 was used to design the flat and ring plastic part.

The processes started by having the product sketching, in this stage, the idea on the basic structure of the part were drawn on a piece of paper in order to overview the part. The second process was the product drafting processing, this was made by using the CAD software to generate the Two Dimensional (2D) drawing as shown in Figure 6(a). In this stage, all drawings must be given dimension in order to constrain the shape from changing. The third step in designing the plastic part was extruding, in other CAD software this step also called padding, and this was done in order to create Three Dimensional Solid Modeling (3D Modeling). Figure 6(b) shows the extruded phase. In the extruded command, the height or thickness of the part should be decided through recommendation wall thickness of the plastic part.

The fourth step are in order to made the plastic part easily removal from the mould, it needed to create the draft angle and filleting at the selected edge around the corner of the plastic part. Figure 6(c) shows the drafting step takes place where the draft surface and normal surface that is need to be chosen.

The purple surface was the face that the product dimension will not change and the draft surfaces were the blue color. Each face has to be selected separately and the draft angle can be dimensioned to the desired value. Figure 6(d) shows the cross section of the part with 5° draft angles were indicated by arrow at wall face of plastic parts.

The last step as show in figure 7(e) was filleting the desired edge or in some CAD software it was called fillet radius, the filleting steps with 5mm radius at four corners on the flat plastic parts and eight corner on the ring plastic parts are done. Fillet can be made on the desired edge and in this case of plastic part, the edge in Figure 6(e) shows the selected edge by the yellow mark.

Finally, the designs for both plastic parts were completed. Figure 6(f) shows the completed flat and ring plastic part. Location of gate on the completed plastic parts will be decided when the structure of plastic injection mould was decided. There two types of plastic injection mould, where two plate mould used side gate system and three plate mould used pin point gate system.

2.2 Insert Mould Design

There are many parts in the slanted glass mould and there are many steps involve in designing each part. However, by using SolidWorks software, the design process can be performed to make all of the mould parts. There are three main parts in the slanted glass mould that is the core insert for flat plastic part, core insert for ring plastic part and the slanted glass.

2.2.1 Core Insert for Flat Plastic Part Design Process

Starting from 2D drawing, there are several command and steps such as 2D drafting, following by extruding to make the main solid body and also some other command such as drafting and filleting that is need to be done in order to complete the 3D solid modelling for certain part that is designed. There are many types of command in SolidWorks that is not covered in this paper however it is still be used in order to model other part of the mould. Figure 7 below shows the core insert that is work as cavity in the plastic injection mould and Table 1 shows the steps and command that is used in SolidWorks.

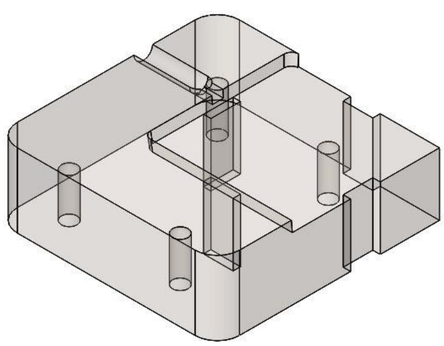


Figure 7 3D model of core insert for flat plastic part

Table 1 Steps to design core insert

	Command	Descriptions
1	Sketch	2D drawing for base (60x60mm)
2	Extrude	Create 3D solid modeling (20mm)
3	Sketch	2D drawing on top plane for cavity
4	Extrude Cut	Cut the area for cavity with draft angle 5° (40x40x3mm)
5	Sketch	2D drawing on cavity plane for flat ejector pin (8x2mm)
6	Extrude Cut	Cut the area for flat ejector pin
7	Sketch	2D drawing on top plane for runner (2.5x15mm)
8	Cut Revolve	Cut the area for runner in semi-cylindrical
9	Sketch	2D drawing for gate on top plane (3x5x2mm)
10	Extrude Cut	Cut the area for gate
11	Fillet	Filleting the edge of cavity area (R2mm)
12	Fillet	Filleting the edge of main body (R6mm)

2.2.2 Core Insert for Ring Plastic Part Design Process

The other part is the core insert ring where ring plastic part is made using this insert. Figure 8 shows the core insert ring while Table 2 shows the steps for designing the core insert ring.

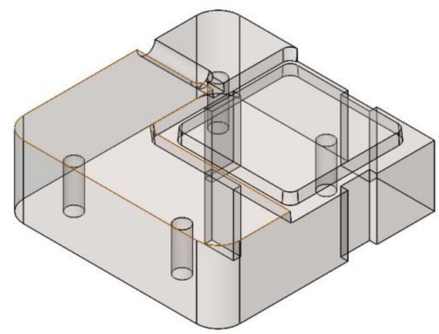


Figure 8 3D model of core insert for ring plastic part

Table 2 Steps to design core insert ring

	Command	Descriptions
1	Sketch	2D drawing for base (60x60mm)
2	Extrude	Create 3D solid modeling (20mm)
3	Sketch	2D drawing on top plane for cavity
4	Extrude Cut	Cut the area for cavity with draft angle 5° (40x40x3mm)
5	Sketch	2D drawing on cavity plane for flat ejector pin (8x2mm)
6	Extrude Cut	Cut the area for flat ejector pin
7	Sketch	2D drawing on top plane for runner (2.5x15mm)
8	Cut Revolve	Cut the area for runner in semi-cylindrical
9	Sketch	2D drawing for gate on top plane (3x5x2mm)
10	Extrude Cut	Cut the area for gate
11	Sketch	2D drawing on cavity plane for ring shape
12	Extrude	Create 3D solid modeling with draft angle 5° (32x32x3mm)
13	Fillet	Filleting the edge of cavity area (R2mm)
14	Fillet	Filleting the edge of main body (R6mm)

2.2.3 Slanted Glass Design Process

The most important part in this slanted glass injection mould is the slanted glass. It works as a window that can helps researcher to visualise the flow of the molten plastic material flow into the cavity. Figure9 shows the slanted glass while Table 3 shows the steps for designing the slanted glass.

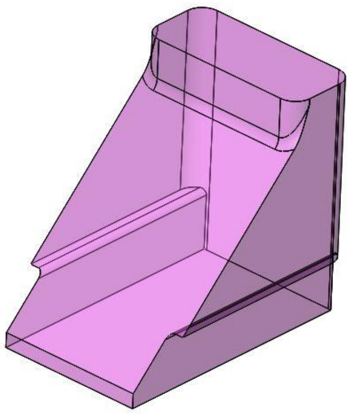


Figure 9 Slanted glass

Table 3 Steps to design slanted glass

	Command	Descriptions
1	Sketch	2D drawing for base (85x60mm)
2	Extrude	Create 3D solid modeling (80mm)
3	Sketch	2D drawing on back plane for cavity
4	Extrude Cut	Cut the area for slot with (20x5mm) for both slot
5	Sketch	2D drawing on side plane for slanted side 45°
6	Extrude Cut	Cut the area for slanted are to place mirror
7	Fillet	Filleting the edge of main side (R6mm)
8	Fillet	Filleting the edge of slot area (R2mm)

Noted that several steps are same because the difference between this two core insert only on step 11 and step 12. The steps for designing the part by using CAD Software are not constraints to these following steps in the table. The designer may choose any command that is appropriate to what they want to do first. However the first two steps are must because it is the basic to form the main 3D solid before modifying it to the desired design.

3.0 MOULD BASE OF SLANTED GLASS

3.1 Slanted Glass Mould Set-Up

In order to design the slanted glass mould base, the main concept of design has been done in a view of a diagram as shown in Figure 10. Front view shows the position of the slanted glass where digital camera no. 1 can capture the flow image from the top view of the product from the deflection of the mirror at the slanted part of the mirror.

Meanwhile, top view in Figure 10 shows the digital camera no. 2 and no. 3 that can capture the flow image from front and side of the product through the glass part. All digital cameras capture the flow image simultaneously during each injection shot.

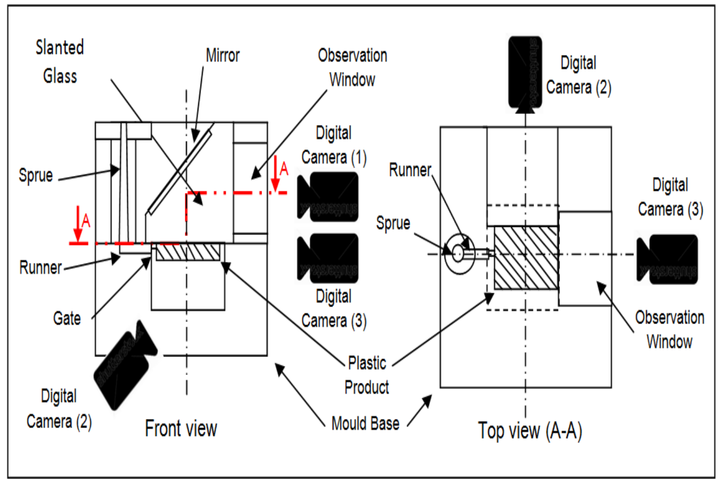


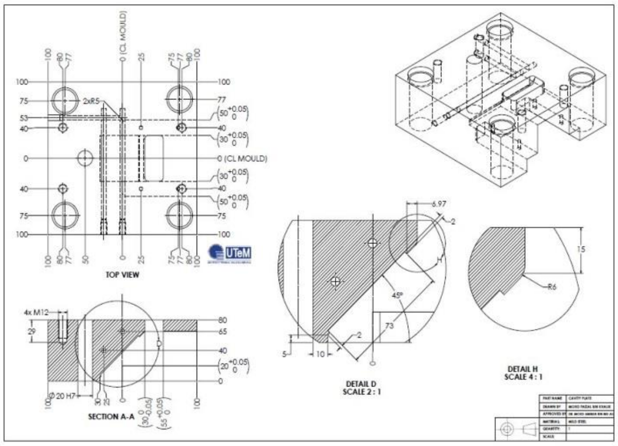
Figure 10 Slanted glass mould diagram

3.2 Stationary / Moveable Mould Design

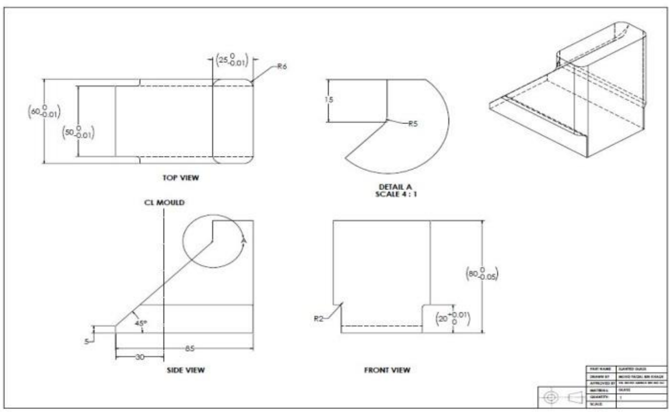
The two-plate mould injection mould design consists of two main plates and that are the core plate and the cavity plate. The cavity plate area is known as stationary side and the core plate area known as the moveable side.

3.2.1 Stationary Side Mould Design

Figure 11(a) shows cavity plate detail drawing and slanted glass detail drawing of stationary side of the slanted glass injection mould. For each process cycle, molten material is shot from the machine through nozzle and go through the sprue. The material is then flow to the cavity side following the shape of the cavity or product. The slanted glass is fixed to the cavity plate by using slot as shown in Figure 11(b).



(a)

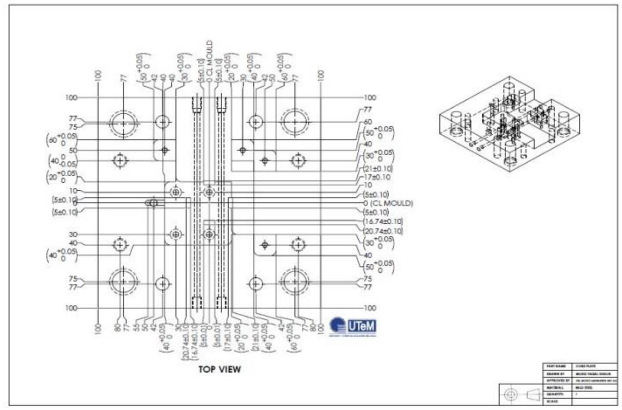


(b)

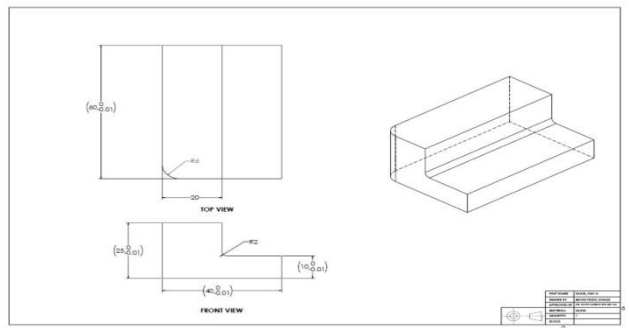
Figure 11 (a) Cavity plate detail drawing and **(b)** Slanted glass detail drawing

3.2.2 Moveable Side Mould Design

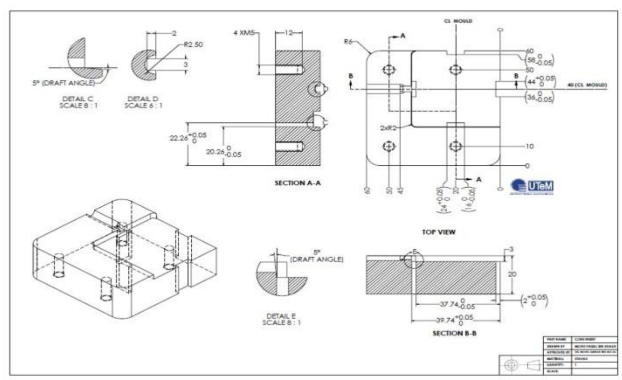
Figure 12(a), (b) and (c) show core plate detail drawing, side glass detail drawing and core insert detail drawing of moveable side of slanted glass injection mould. Core plate is the main part where the insert is fixed at the middle of it. The insert decide the shape of the finish product. During the injection process taken place, the molten material is flowing through sprue at the cavity plate. It is then flow through the runner and gate at the core insert and after that flow inside the cavity space. While the material is filling up the cavity space, digital camera no. 1, no. 2 and no. 3 captured the flow material through the slanted glass and the side glass.



(a)



(b)



(c)

Figure 12 (a) Core plate detail drawing **(b)** side glass detail drawing and **(c)** core insert detail drawing

4.0 PROCUREMENT AND ASSEMBLY DRAWING

4.1 Procurement of Mould Base, Mould Plate and Mould Standard Parts

Procurement process is essential in mould making which the various mould plates were purchased by referring to the list of assembly drawing as shown in Table 4. During purchasing of raw material the size of mould plate was added from the original size stated at the assembly drawing. Normally in mould making practices, the thickness of mould plate was added

1mm per side, meanwhile 5 mm width and length. The reason the size of plate was added, because of the raw material need for squaring process which raw material received was in rough surface and not full square shape. For example for core insert part no. 9, the ordering size is 65mm x 65mm x 22mm. The mould standard parts were purchasing follow the items found from mould supplier catalogue such as company from Misumi and DME. While for mould base, the mould base company available was Futaba and Lung Kee Group (LKM).

Table 4 List of procurement.

No.	PART NAME	REMARKS	MATERIAL	QTY.
1	CAVITY PLATE	200 x 200 x 80	MILD STEEL	1
2	MIRROR	73 x 60 x 2		1
3	SLANTED GLASS	85 x 80 x 60	GLASS	1
4	SLOT LEFT	100 x 25 x 20	MILD STEEL	1
5	SLOT RIGHT	100 x 25 x 20	MILD STEEL	1
6	TOP PLATE	250 x 250 x 25	MILD STEEL	1
7	LOCATING RING	Ø125 x 8	MILD STEEL	1
8	CORE PLATE	200 x 200 x 50	MILD STEEL	1
9	CORE INSERT	60 x 60 x 20	STAVAX	1
10	FLAT EJECTOR PIN	Ø15 x 122	MILD STEEL	4
11	GLASS_FLAT A	60 x 40 x 25	GLASS	1
12	GLASS_FLAT B	50 x 40 x 25	GLASS	1
13	BACKUP CORE PLATE	200 x 200 x 30	MILD STEEL	1
14	GUIDE PINS	Ø25 x 130	MILD STEEL	4
15	RETURN PIN	Ø17 x 125	MILD STEEL	4
16	SPACER BLOCK 1	200 x 40 x 60	MILD STEEL	1
17	SPACER BLOCK 2	200 x 40 x 60	MILD STEEL	1
18	TOP EJECTOR PLATE	200 x 118 x 13	MILD STEEL	1
19	BOTTOM EJECTOR PLATE	200 x 118 x 15	MILD STEEL	1
20	BOTTOM PLATE	250 x 250 x 25	MILD STEEL	1

4.1.2 Assembly Drawing

After all the part has been fabricated, they have to be assembled together by following the assembly drawing in Figure 13.

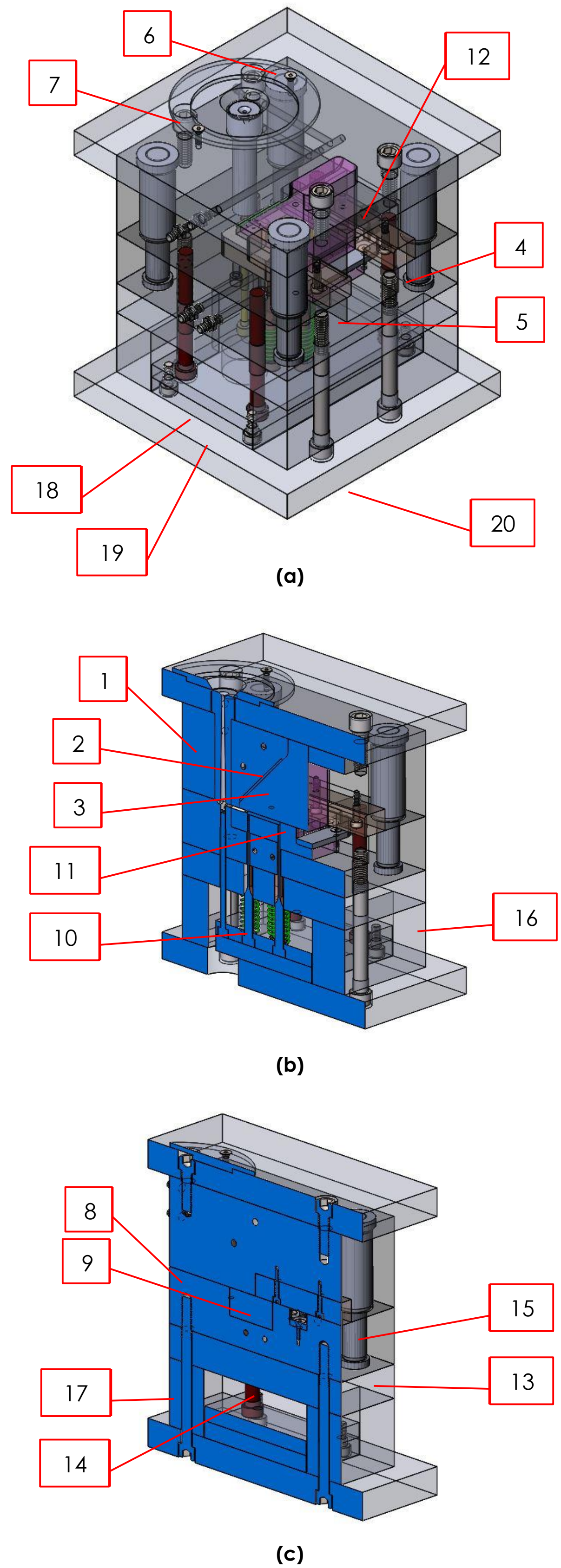


Figure 13 (a) Isometric view of mould assembly **(b)** Section A-A of mould assembly **(c)** Section B-B of mould assembly

The assembly process must follow exactly the number of placement of the part otherwise the mould could not operate normally and also might cause an error in functioning.

The fabricated mould will be used to visualize the flow pattern of molten material moving inside the mould cavity area. The flow of molten plastic material is investigated and analysed to see whether it can affected the product quality i.e., weld line, sink mark and air burble etc. when it fills the cavity area.

5.0 CONCLUSION

The design of product and tooling of slanted glass injection mould was carried out. A two-plate injection plastic mould having a new method of embedded slanted glass at mould cavity side was designed where it can be used to visualize the flow of molten plastic material moving inside the mould cavity during the process. Flow of the molten plastic from gate into the mould cavity can be observed through slanted glass; front and side glass using digital camera and the image of plastic flow taken can be analyzed through the visualization and compared with modeling simulation. The design process for both plastic part and slanted glass mould was carried out by using Solidworks software. The flat plastic product and ring plastic product were designed with the consideration of corner design and wall thickness while the slanted glass mould was designed with consideration of feeding system, fitting system and mould cooling system to ensure the finished mould can be operated well. The slanted glass injection mould is designed carefully because in actual fabrication of mould need high investment and any error of mould design should be avoided before actual plastic injection mould is machined and fabricated. By having the finished design of slanted glass plastic injection mould, the fabrication of actual plastic injection mould of slanted glass can be started.

Acknowledgement

The authors gratefully acknowledge to Universiti Teknikal Malaysia Melaka (UTeM) for supported this project under FRGS/1/2014/TK01/FKP/02/F00224.

References

[1] Ahmadzai, A., Behrave, A.H., Sarabi, M.T. and Shahi, P. 2014. Visualization of Foaming Phenomena in Thermoplastic Injection Molding Process. *Journal of Cellular Plastics.* 50(3): 279-300.

[2] Amran, M. A., Hadzley, M., Amri, S., Izamshah, R., Hassan, A., Samsi, S. and Shahir, K. 2010. Optimization of Gate, Runner And Sprue in Two-Plate Family Plastic Injection Mould. *AIP Conference Proceedings.* 1217: 309-313.

[3] Bress, T. J. and Dowling, D. R. 1998. Visualization of Injection Molding. *Journal of Reinforced Plastics and Composites.* 17(15): 1374 -1381.

[4] Bociga, E. and Jaruga, T. 2007. Experimental Investigation of Polymer Flow in Injection Mould. *Archives of Materials Science.* 28(3): 165-172.

[5] Kanetoh, Y. and Yokoi, H. 2012. Visualization Analysis of Resin Behavior Around a Flow Front Using a Rotary Runner Exchange System. *International Polymer Processing.* 3(3): 310-317.

[6] Amran, M., Salmah, S., Zaki, M., Izamshah, R., Hadzley, M., Sivarao, Shahir, M. and Amri, M. 2014. The Effect Of Pressure On Warpage of Dumbbell Plastic Part in Injection Moulding Machine. *Advanced Materials Research.* 903: 61-66.

[7] Jou, Y. T., Lin, W. T, Lee, W.C. and Yeh, T. M. 2014. Integrating The Taguchi Method And Response Surface Methodology for Process Parameter Optimization of The Injection Molding, *Applied Mathematics & Information Sciences.* 8(3): 1277-1285.

[8] Amran, M., Salmah, S., Faiz, A., Izamshah, R., Hadzley, M., Manshoor, B., Shahir, M. and Amri, M. 2014. Effect of Injection Moulding Machine Parameters on The Warpage by Applying Taguchi Method. *Applied Mechanics and Materials,* 699: 20-25.

[9] Amran, M., Izamshah, R., Hadzley, M., Shahir, M., Amri, M., Sanusi, M. and Hilmi, H. 2015. The effect of binder on mechanical properties of kenaf fibre/polypropylene composites using full factorial method, *Applied Mechanics and Materials.* 695**:** 709-12

[10] Rosfadzlin H. Z. A., Taufik, Sivaraos, S., Ghazali, M., Rosidah, J. and Amran, M. A. M. 2010. Design Simulation on Fatigue Life of Fiber Reinforced Polymer Composites. *Applied Mechanics and Materials.* 761: 47-51.

[11] John P. B. 2007. *Runner and Gating Design Handbook*. Hanser Gardner Publication

[12] Nagel, R., Sherman R., and Beaumont J. P. 2003. *Successful Injection Molding – Process, Design, and Simulation.* Hanser Gardner Publication

[13] Douglas M. B. 1998. *Plastic Injection Molding.* Society of Manufacturing Engineers

[14] Amran, M., Izamshah, R., and Amri, M. 2007. Study The Machining Process on Acrylic Material of Mould Base in Plastic Injection Mould. Proceeding of *International Conference on Engineering Technology (ICET2007).*

[15] Menges, G. Mohren, P. 1993. *How To Make Injection Moulds. 2nd Edition,* Hanser Gardner Publication

[16] Potsch, G. Michaeli, W. 1995. *Injection Molding: An Introduction.* Carl Hanser Verlag.