Design consideration for design a flat and ring plastics part using Solidworks software

2015 IOP Conf. Ser.: Mater. Sci. Eng. 100 012050

(http://iopscience.iop.org/1757-899X/100/1/012050)

IOP Conf. Series: Materials Science and Engineering **100** (2015) 012050 doi:10.1088/1757-899X/100/1/012050

Design consideration for design a flat and ring plastics part using Solidworks software

M A M Amran[1], K M Faizal[1], M S Salleh[1], M A Sulaiman[1], and E Mohamad[1]
[1]Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka, 76100 Durian Tunggal, Melaka, Malaysia

E-mail: mohdamran@utem.edu.my

Abstract. Various considerations on design of plastic injection moulded parts were applied in initial stage to prevent any defects of end products. Therefore, the objective of this project is to design the plastic injection moulded part by taking consideration on several factors such as draft angle, corner radius and location of gate. In this project, flat plastic part, ring plastic part, core inserts for flat and ring plastic part were designed using SolidWorks software. The plastic part was drawn in sketching mode then the 3D modeling of solid part was generated using various commands. Considerations of plastic part such as draft angle and corner radius with location of gate was considered in the design stage. Finally, it was successfully designed the two plastic parts with their respectively insert by using SolidWorks software. The flat plastic part and ring plastic part were designed for the purpose for future researches for study the weld lines, meld lines, air trapped and geometrical size of the product. Thus, by designing the flat plastic part and ring plastic part having core insert on each part, the completed mould design of two plate mould can be considered. This is because, plastic injection parts are needed to be designed properly in order to neglect any defect when the mould was made.

1. Introduction

Over recent years, there are many types of Computer Aided Design (CAD) software available around the world. These softwares lead the imagination and creativity of engineers to interpret and deliver their good ideas using this kind of medium. Then, others people can understand well what and how they want to execute their ideas. CAD softwares are used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing [1]. CAD software outputs are often in the form of electronic files for print, machining, or other manufacturing operations [2].

In mechanical design it is known as mechanical Design Automation (MDA) or Computer-Aided Drafting (CAD), which includes the process of creating a technical drawing with the use of computer software [3]. CAD softwares for mechanical design use either vector-based graphics to describe the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances. There are various types of CAD softwares available in the market nowadays such as Solidworks, AutoCAD, CATIA, Pro Engineering, etc., not to mention with various capability to analyze the after design product.

In designing and producing the plastic injection molding parts, it is hard by only using the conventional method and by not using Computer Aided Design (CAD) softwares. However, creating or designing the part is not enough, injection molding parts are affected by many factor such as corner design, draft angles, the location of the edge gate and those must be consider in order to get the best design without neglecting the main purpose of the part itself. Sharp corner will block the resin flow when molding, which will become a cause of the flow marks. Also, the stress will concentrate on the sharp corner part, so it likely to be the cause of deterioration in strength such as notch effect and residual strain. In this project, two plastic parts has been selected for design. Selections of two types of these plastic parts are based on the wall 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.

2. Methodology

2.1. Experimental Procedure

There are several steps involving plastics part design from sketching until 3D modeling in order to achieve the finish completed 3D solid modeling. Figure 1 shows the flow chart in order to design the plastic part. In this process, Solidworks 2013 software was used to design the flat and ring plastic part.

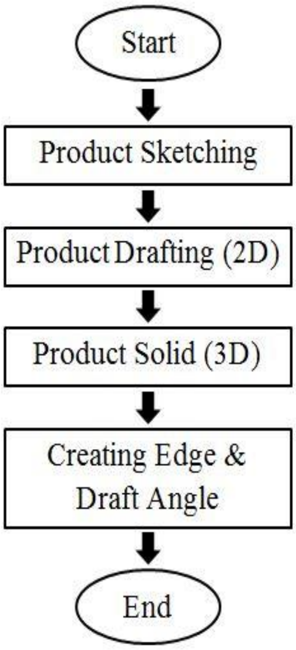

Figure 1. Flow chart to design 3D solid plastic part.

2.1.1. 2D Drawing. 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 2.

IOP Conf. Series: Materials Science and Engineering **100** (2015) 012050 doi:10.1088/1757-899X/100/1/012050

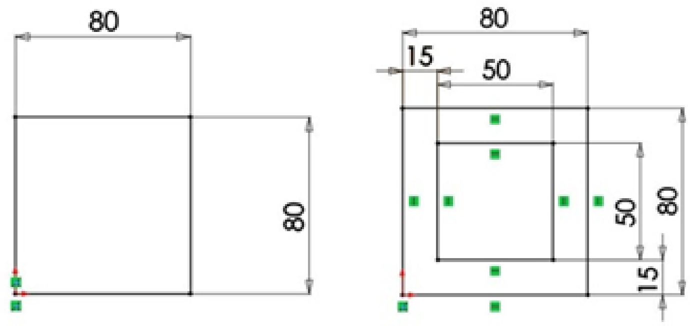

Figure 2. 2D drafting of flat and ring part.

2.1.2. 3D Solid Modeling. 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 3 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.

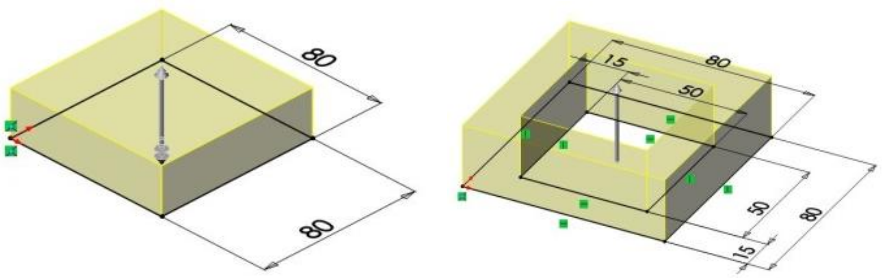

Figure 3. Extruded process for flat and ring part.

The next step 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 4 shows the drafting step takes place where the draft surface and normal surface that is need to be chosen.

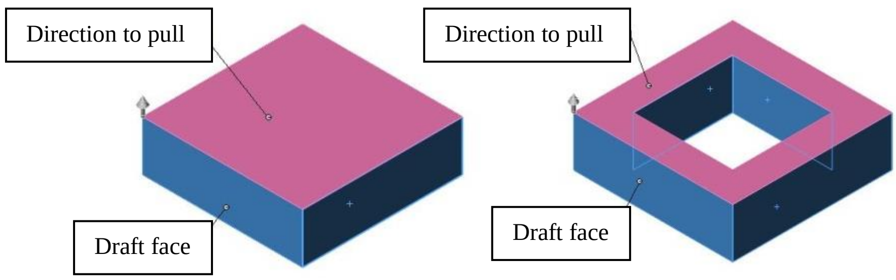

Figure 4. Draft surface and normal surface in drafting.

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 5 shows the cross section of the part with 5° draft angles were indicated by arrow at wall face of plastic parts.

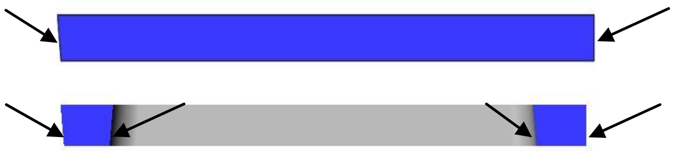

Figure 5. Draft angle 5° at the flat and ring part.

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

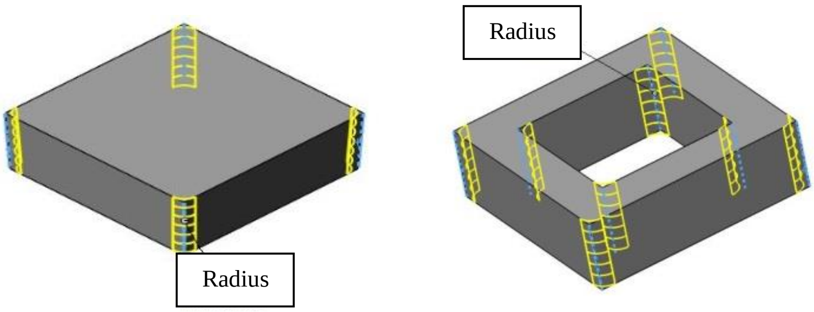

Figure 6. Fillet on the desired edge on flat and ring part.

2.1.3. Completed Part 3D Solid Modeling. Finally, the designs for both plastic parts were completed. Figure 7 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 mold used side gate system and three plate mould used pin point gate system.

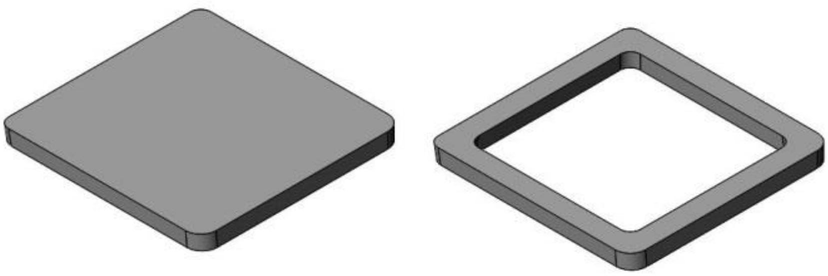

Figure 7. Flat plastic part and ring plastic part.

3. Steps for Modeling Core Insert for Flat Plastic Part

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 modeling 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 8 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.

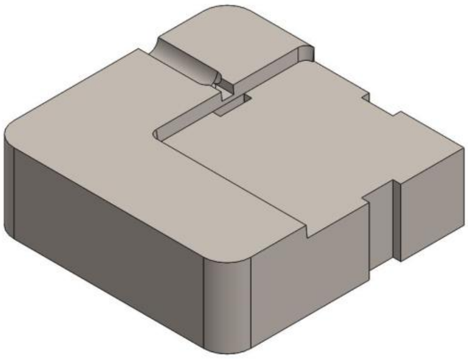

Figure 8. 3D model of core insert for flat plastic part.

Table 1. Steps to design core insert.

	Command	Descriptions
Step 1	Sketch	2D drawing for base (60x60 mm)
Step 2	Extrude	Create 3D solid modeling (20 mm)
Step 3	Sketch	2D drawing on top plane for cavity
Step 4	Extrude cut	Cut the area for cavity draft angle 5º (40x40x3 mm)
Step 5	Sketch	2D drawing on cavity plane for flat ejector pin (8x2 mm)
Step 6	Extrude cut	Cut the area for flat ejector pin
Step 7	Sketch	2D drawing on top plane for runner (2.5x15 mm)
Step 8	Cut revolve	Cut the area for runner in semi-cylindrical
Step 9	Sketch	2D drawing for gate on top plane (3x5x2 mm)
Step 10	Extrude cut	Cut the area for gate
Step 13	Fillet	Filleting the edge of cavity area (R2 mm)
Step 14	Fillet	Filleting the edge of main body (R6 mm)

4. Steps for Modeling Core Insert Ring for Ring Plastic Part

The other part is the core insert ring where ring plastic part is made using this insert. Figure 9 shows the core insert ring while table 2 shows the steps for designing the core insert ring.

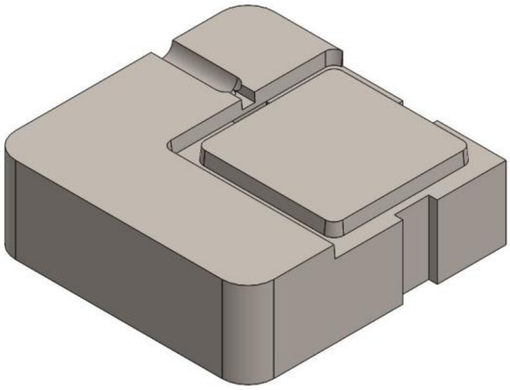

Figure 9. 3D model of core insert ring for ring plastic part.

Table 2. Steps to design core insert ring.

	Command	Descriptions
Step 1	Sketch	2D drawing for base (60x60 mm)
Step 2	Extrude	Create 3D solid modeling (20 mm)
Step 3	Sketch	2D drawing on top plane for cavity
Step 4	Extrude cut	Cut the area for cavity draft angle 5º (40x40x3 mm)
Step 5	Sketch	2D drawing on cavity plane for flat ejector pin (8x2 mm)
Step 6	Extrude cut	Cut the area for flat ejector pin
Step 7	Sketch	2D drawing on top plane for runner (2.5x15 mm)
Step 8	Cut revolve	Cut the area for runner in semi-cylindrical
Step 9	Sketch	2D drawing for gate on top plane (3x5x2 mm)
Step 10	Extrude cut	Cut the area for gate
Step 11	Sketch	2D drawing on cavity plane for ring shape
Step 12	Extrude	Create 3D solid modelling with draft angle 5º (32x32x3 mm)
Step 13	Fillet	Filleting the edge of cavity area (R2 mm)
Step 14	Fillet	Filleting the edge of main body (R6 mm)

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.

5. Discussion

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.

5.1. 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 the nominal wall thickness shows in figure 10. The result is a longer cooling time accompanied by a risk of sink marks and warping due to differential shrinkage [4].

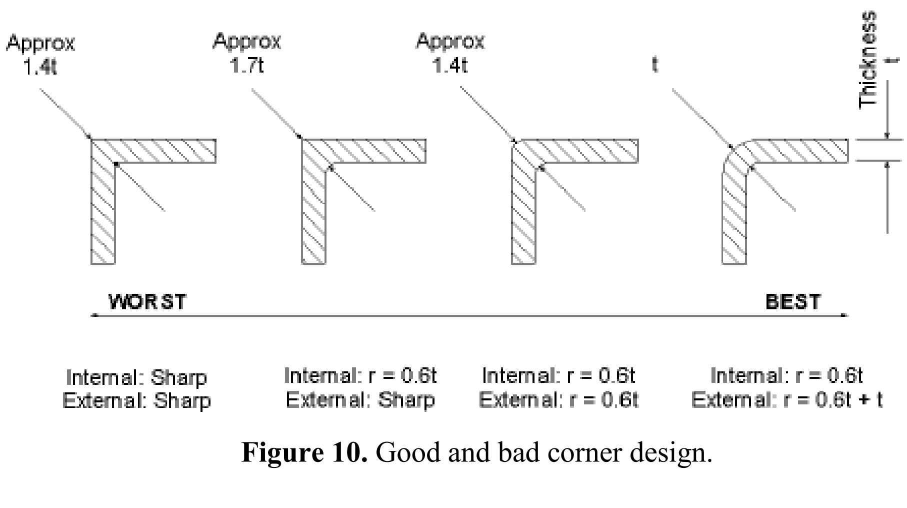

Figure 10. Good and bad corner design.

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 [5]. 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 [6].

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 minimising flow-front break-up.

5.2. 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 [7]. 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 effect that will cause ejection problems. Figure 11 shows example of ribs on the plastic parts.

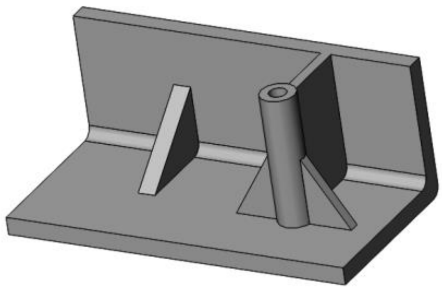

Figure 11. Example of ribs on the plastic parts.

5.3. 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. 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 mouldings 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. In practice, there must be some variation in thickness to accommodate and fulfil functions or aesthetics. It is very important to keep this variation to a minimum. Figure 12 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 [8].

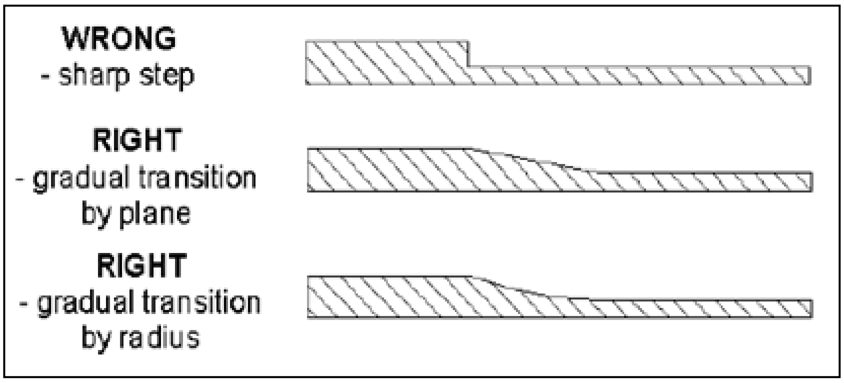

Figure 12. Gradual transitions between thick and thin sections.

6. Conclusion

The design consideration for designing a flat and ring plastic part using SolidWorks software was carried out. This project was successfully design the flat plastic part, ring plastic part, core insert for flat plastic part and ring plastic part using SolidWorks software. Flat plastic part was added external draft angle at the four edges meanwhile the ring plastic part was added to internal and external draft angle with both angle of 5°. The edge gate location was considered for flat plastic part and ring plastic part. By taking the consideration to design the plastic part, the new completed of two plate injection mould will be proposed.

Acknowledgements

The authors would like to thank Faculty of Manufacturing Engineering, Universiti Teknikal Malaysia Melaka for providing facilities for this research to be conducted successfully. This research is funded by the Ministry of Higher Education through Fundamental Research Grant Scheme (FRGS/1/2014/TK01/FKP/02/F00224).

References

[1] Rao K M, Narayan K L 2008 *Computer Aided Design and Manufacturing* PHI Learning)
[2] Farin G E, Hoschek J, Kim M-S 2002 *Handbook of computer aided geometric design* Elsevier)
[3] Madsen D A, Madsen D P, Turpin J L 2012 *Engineering drawing & design* (Clifton Park, NY: Delmar, Cengage Learning)
[4] Suzuki M, Ali M A M, Okamoto K, Taniike T, Terano M, Yamaguchi M 2009 *Advances in Polymer Technology* **28** 185-91
[5] Amran M, Izamshah R, Hadzley M, Shahir M, Amri M, Sanusi M, et al. 2015 *Applied Mechanics and Materials* vol 695 Trans Tech Publ) p. 709-12

[6] Amran M A, Hadzley M, Amri S, Izamshah R, Hassan A, Samsi S, et al. 2007 *International Conference on Advancement of Materials and Nanotechnology:(ICAMN—2007)* (Langkawi Island, Malaysia) vol 1217 AIP Publishing) p. 309-13
[7] Amran M, Salmah S, Faiz A, Izamshah R, Hadzley M, Manshoor B, et al. 2015 *Applied Mechanics and Materials* vol 699 Trans Tech Publ) p. 20-5
[8] Amran M, Salmah S, Zaki M, Izamshah R, Hadzley M, Kasim M S, et al. 2014 *Advanced Materials Research* vol 903 Trans Tech Publ) p. 61-6