Ideal practice while designing injection molded plastic part

[1]Gupta Narendra Somnath, [2]Prof. R. K. Agrawal

[1]Student of ME in Product Design and Development, YTCEM -Bhivpuri road-Karjat, Maharashtra, India
[2]HOD, Mechanical engineering Department, YTCEM -Bhivpuri road-Karjat, Maharastra, India

***Abstract*—This document helps to explain the basic and important guideline while designing a plastic part. If this ideal practice refers while designing part then the designer will deliver a product which is suitable as per the manufacturing and production point of view.**

Key words- **Injection molding, Draft angle, Sink marks, Voids, Undercut, Coring, Straight-pull, Gussets, air-trap, Fillet**

I. Introduction

It is necessary to understand the limitation and the criticality of the process before designing a product which is going to be developed by that process. If there is a plastic product which is going to develop by Injection molding, then that product should be designed by keeping the ideal practice of that process. If a designer developed a product with better aesthetic and functional point of view, only than sometime it is too complicated, costly, time consuming to produce that product or equal to impossible. So it is better to explore the ideal practice guideline of injection molded plastic product design before designing it. This document gives you the ideal practices. Guideline for wall-thickness, draft-angle, fillet, chamfer, bosses, sharp-corner, parting-line, under-cut etc. In this some examples are given to explain the designing assumptions.If designed studies this kind of the document before designing a plastic part then it is very helpful to be delivered a practically feasible product which can be produced with very less effort.

Almost anything can be produced by injection molding in plastic. Injection molding is one of the most widely used manufacturing processes in the world today. Look around you and you’ll probably see dozens of injection molded parts in your wallet, kitchen, car and office. Such a widely used manufacturing process must have a fair few advantages for getting things done this way, right? Let’s have a look at some of the advantages and disadvantages of injection molding.

Advantages of Injection Molding

- Fast Production-Injection molding can produce an incredible amount of parts per hour. It’ll depend on how many impressions (part molds) are in your tool, but you’re looking at something between 15-30 seconds for each cycle time.
- Material and Color Flexibility-Once you have a tool made, without lots of difficulty, you can change the material and color of the part that you’re producing.
- Labor Costs Low-A self-gating, automatic tool runs on an injection molding machine without very much difficulty at all. Your parts can be readied with little or no labor on top of the production. Low Waste
- Most plastics recycle – we grind up all of the waste that we can and reuse it, thus reducing our waste.

Disadvantages of Injection Molding

- High initial tooling cost
- Part design limitation.

II. Problem identification and definition with the injection molding part design

After designing a part there may be needed to make few changes to parts so that can make or decide on a different manufacturing technique. The most important thing to realize is that a mold tool is made from two halves that need to pull apart and the part needs to be able to be released from the tool. This is simple, but massive. It has all sorts of ramifications down the line in terms of tool design like:

- If there is no draft to part then it is difficult to release the part of the tool.
- Undercut requires side-core with is cost effective.
- Uniform wall thickness required for easy flow of plastic material.
- Sometime Sharp-edges are very difficult to fill and manufacturing in the tool.
- Limitation of Straight Pull.

From the above review the problem can be defined as with the injection molded part design there is some limitation which need identify at the time of the product design which will help to save the cost and time.

III. IDEAL PRACTICE DESIGN GUIDELINE

3.1. WALL-THICKNESS

Wall thickness strongly influences many key part characteristics, including mechanical performance and feel, cosmetic appearance, moldability, and economy. The optimum thickness is often a balance between opposing tendencies, such as strength versus weight reduction or durability versus cost. Give wall thickness careful consideration in the design stage to avoid expensive mold modifications and molding problems in production. Parts should be designed with a minimum uniform wall thickness consistent with part function and mood filling considerations. Thinner the wall faster the part cools, and the cycle times are short, resulting in the lowest possible part costs. Also, thinner parts weight loss, which results in smaller amounts of the plastic used per part which also results in lower part costs.

3.2. NEED FOR UNIFORM WALL

Thick sections cool slower than thin sections. The thin section first solidifies, and the thick section is still not fully solidified. As the thick section cools, it shrinks and the material for the shrinkage comes only from the unsolidified areas, which are connected, to the already solidified thin section. This builds stresses near the boundary of the thin section to thick section. Since the thin section does not yield because it is solid, the thick section (which is still liquid) must yield. Often this leads are twisted. If this is severe enough, the part could even crack.

- The material shrinkage increases with the thickness. An excessive thickness may result in sink marks or voids.
- It is also recommended to avoid having a thin area surrounded by a thick perimeter section as it could result in air-trap

3.3. WHAT IF YOU CANNOT HAVE UNIFORM WALLS, (DUE TO DESIGN LIMITATIONS)?

If design limitations make it impossible to have uniform wall thicknesses, the change in thickness should be as gradual as possible.

- Coring is a method where plastic is removed from the deck area, which helps to keep Wall sections uniform, eliminating the problem altogether. Wall sections uniform, eliminating the problem altogether.
- Gussets are support structures that can be designed into the part to reduce the possibility of warping.

3.4. DRAFT ANGLE

Mold drafts facilitate part removal from the mold. The draft must be at an offset angle that is parallel to the mold opening and closing. The ideal draft angle for a given part depends on the depth of the part in the mold and its required end-use function.

3.5. FILLET/RADII

Sharp corners greatly increase stress concentration, which, when high enough, can lead to part failure. Sharp corners often come about in non-obvious places, such as a boss attached to a surface, or a strengthening rib. In addition to reducing stress, the fillet radius provides a streamlined flow path for the molten plastic, resulting in an easier fill of the mold. A bigger radius should be used if the part design allows.

3.6. BOSSES

Bosses are used for the purpose of registration of mating parts or for attaching fasteners such as screws or accepting threaded inserts (molded-in, press-fitted, ultrasonically or thermally inserted). Wall thicknesses for bosses should be less than 60 percent of the nominal wall to minimize sinking. The base radius should be a minimum of 0.25 X thickness. Bosses can be strengthened by incorporating gussets at the base or by using connecting ribs attaching to nearby walls. Bosses should not be located along any edge. If rigidity is required the bosses can be linked to the edge by ribs.

3.7. RIBS

Ribs in plastic part improve bending stiffness (relationship between load and part deflection) of the part and increases rigidity. It also enhances mouldability as they hasten melt flow in the direction of the rib. Ribs are placed along the direction of maximum stress and deflection on nonappearance surfaces of the part. Proper rib design involves five main issues: thickness, height, location, quantity, and mouldability. Consider these issues carefully when designing ribs.
Height-The height of a rib should be limited to less than three times its thickness. It is better to use multiple ribs to increase bending stiffness than to use one very tall rib.
Thickness-Rib thickness should be less than the wall thickness to minimize sinking effects. The recommended rib thickness should not exceed 60 percent of the nominal thickness. Plus, the rib should be attached with corner radii as generous as possible.
Location-The rib orientation is based on providing maximum bending stiffness. Depending on the orientation of the bending load, with respect to the part geometry, ribs oriented one way increase stiffness. If oriented the wrong way there is no increase in stiffness.
At rib intersections, the resulting thickness will be more than the thickness of each individual rib. Coring or some other means of removing material should be used to thin down the walls to avoid excessive sinking on the opposite side.
Mouldability- Draft angles for ribs should be minimum of 0.25 to 0.5 degrees of draft per side.

3.8. Sharp corner

Avoid sharp corners as it could lead to high stress concentration, or in some case create air traps. Below picture (right) shows the stress concentration as a function of the radius to thickness ratio: a ratio of approximately 0.15 gives a good compromise between performance and appearance.

3.9. Thermal Heat of Expansion

Most of the plastics have a higher coefficient of thermal expansion than metals. Hence a plastic part that is assembled on a metal part can show some excessive stress at its fixing points due to the difference in thermal expansion. On the long term this can lead to some damage. This problem can be addressed by using some slots rather than round holes.

3.10. HOLE

Distance of a hole from the side wall as well as from the edge of another hole should be at least 2 times the wall thickness. To avoid deflection of core pins due to molding pressures, depth of blind holes is restricted to 2-4 times the diameter.

IV. General guideline

4.1. Textures and lettering

Texture and lettering can be molded on the surfaces, as an aesthetic aid or for incorporating identifying information, either for end users or factory. Texturing also helps hide surface defects such as knit lines, and other surface imperfections. The depth of texture or letters is somewhat limited, and extra draft needs to be provided to allow for mold withdrawal without marring the surface. Draft for Texturing is somewhat dependant on the mold design and the specific mold texture. Guidelines are readily available from the mud texture suppliers or mold builders. As a general guideline, 1.5° min. Per 0.025mm (0.001 inch) depth of texture needs to be allowed for in addition to the normal draft.

4.2. Convert metal part to the plastic part

How to convert metal part in plastic? There is no direct method or rule. The metal part is design according the operation by which it is made, while converting we have to be careful from where we can remove the material and where to add the material but without disturbing the function and the strength of the part.

4.3. Insert molding

Inserts used in plastic parts provide a place for fasteners such as machine screws. The advantage of using inserts is that they are often made of brass and are robust. They allow for a great many cycles of assembly and disassembly. Inserts are installed in Injection molded parts using one of the following methods: ULTRASONIC INSERTION, THERMAL INSERTION & MOULDED-IN.

Among the above “molded-In” is most popular and practice. To mold inserts into place during the molding cycle, core pins are used to hold the inserts. The injected plastic completely encases the insert, which provides excellent retention. This process may slow the molding cycle because inserts have to be hand loaded, but it also eliminates secondary operations such as the ultrasonic and thermal insertion methods. Finally, for high volume production runs, an automatic tool can load the inserts, but this increases the complexity and cost of the mold.

4.4. Straight-pull

Design Parts that qualify for Injection Molding must be designed as straight-pull parts. A part made with a straight-pull mold is designed such that when the two halves of the mold pull straight away from each other, there is no mold metal that wants to pass through the part plastic (an impossible, ‘die locked’ situation). Undercuts on the part require mold pieces to pull out sideways, perpendicular to the direction of pull. These are called side actions. Parts with undercuts are not available within the Rapid Injection Molding process. However, undercuts are easily produced using either Low-Volume Injection Molding or Production.

4.5. Undercuts:

In plastic injection molding industry, it refers to part features that prevent straight ejection at the parting line, which cause much mold complexity and lead to higher mold construction and maintenance costs. Whenever it's possible, redesign the part to avoid undercuts. Minor part design changes can often eliminate undercuts in the mold. For example, adding through holes can give access to the underside of features that would otherwise be undercuts, simple modifications enable the mold to form a hole in the sidewalk rather than with a side-action mechanism

V. Conclusion

Starting with a good plastic part design by following above explain design best practices has a much higher return on the investment than a poor design. A good design saves time and money.

These best practices are basic principle of the modeling DFM. These practices look very simple, but they are very effective. Sometime the selection of the tool manufacturing process depends on these basic ideal practices. For any design engineer, before start designing he should go through with the ideal practices of the plastic part design.

REFERENCE

[1] Eugene M. Whimore 1998 “Standard and Practices for injection molder guideline”
[2] Clive Maier (2004) “Design Guides for Plastics”.
[3] Kalpit Jain, Deepak Kumar, (2013) paper publishing on “Plastic Injection Molding with Taguchi Approach - A Review”
[4] James M. Margolis’s “Engineering Plastics Handbook 2006” McGraw-Hill Education
[5] Sanjay K Nayak, Pratap Chandra Padhi, Y. Hidayathullah “Fundamentals of plastic mold design” Tata McGraw-Hill Education
[6] Robert A. Malloy “Plastic part design for injection molding” by SPE Books from Hanser Publishers
[7] “Designing Plastic Parts for Assembly” 7th Edition by Paul A. Tres SPE Books from Hanser Publishers
[8] “The Complete Part Design Handbook: 'For Injection Molding of Thermoplastics” by E. Alfredo Campo
[9] http://www.efunda.com/designstandards/plastic_design
[10] http://www.exothermic.comexothermic
[11] “Ten common mistakes of injection molded plastic part design” By Heppner molds.
[12] Jim Comb, Stratasys, Inc (2003) in the white paper of “How to Design Your Part for Direct Digital Manufacturing”