

Solution Homework #2: Injection Molding

**2.008 Design and Manufacturing II
Spring 2004**

**Out: February 18th
Due: February 25th**

Problem 1:

What are the design considerations in replacing a metal container for carbonated beverages with plastic bottles? Think about the functional requirements and explain how the design of a plastic container differs from that of a metal can.

Solution

While the functional requirements, naturally, are almost the same for metal and plastic containers (remember: functional requirements are “what you want” and solution-neutral), the design parameters to fulfill the functional requirements may differ.

In particular the following differences can be stated:

- Since plastic is a cheap material and less dense than metal, the urge for a material-saving design is not as strong as when using aluminum.
- Special attention has to be paid to the permeability of plastic. In order to conserve the gas for a long period of time (less than 15% loss in 120 days, from the lecture), a special processing with low permeability plastic is necessary.
- Plastic is not as strong as metal. The wall thickness can be increased, small corrugation or, better, a wavy structure (like small coke bottles) can be used. The bottom is usually thick to provide rigidity and a save stand.
- Because plastic is easier to form into complicated shapes than cans, it became more feasible to make them more aesthetically and ergonomically pleasing; hence, soda in bottles again, but now with grippy sides. Also scaling up the size is feasible.
- In principle, plastic didn't eat that much of Al cans, but replaced most of the glass bottles.

Problem 2:

Consider the injection molded L-bracket made of Polycarbonate shown in Figure 1. You are asked to find the mold design with the parting line that leads to the minimum possible clamping force to mold the part.

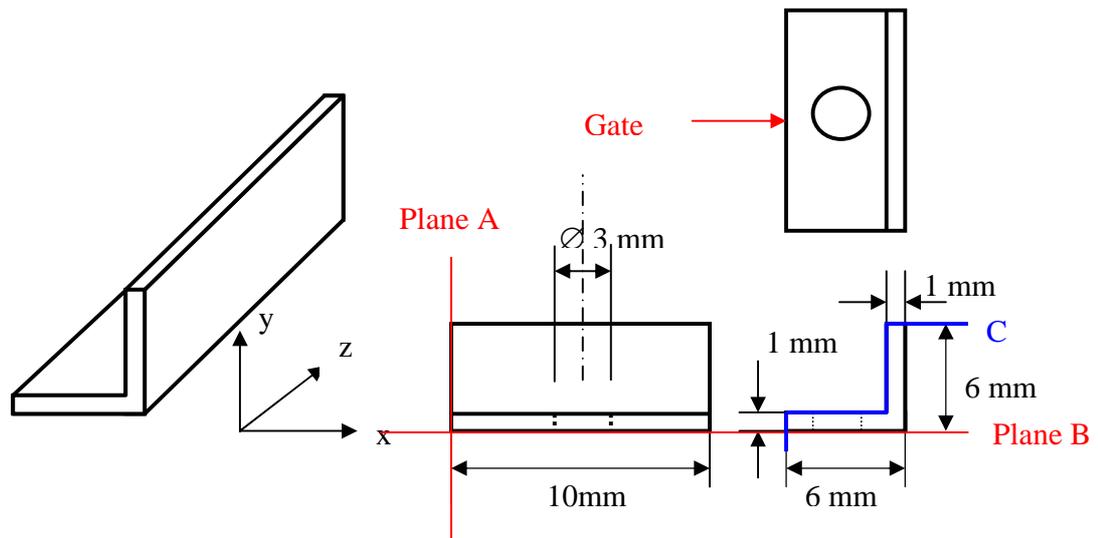


Figure 1: L-bracket part drawings

- a) What would be the minimum clamping force required for this part? Indicate the location of gate and the location of parting line/plane for the minimum clamping force design. Assume that the peak injection pressure is 6,000 psi. Also discuss the potential problems in producing the part with this design.
- b) You soon realize that the minimum clamping force is not as important as the quality of the part and productivity of the process (as long as the machine has enough clamping force). i) What would be a better position of the parting line/plane and gate and direction of injection molding? Why? ii) What is the clamping force required for the new design?

Solution a)

The minimum cross-sectional area projected on the parting plane A will require the minimum clamping force.

$$A_{\min} = 6^2 - 5^2 = 11 \text{ mm}^2$$

$$F_{\min} = P \times A_{\min} = 6000 \times 11 / (25.4)^2 = 102.3 \text{ lbs} = 0.05 \text{ ton}$$

However, this parting line is not a good choice due to the following potential problems.

Potential Problems

- a) The hole on the vertical wall from the parting plane needs a sliding core, which makes it costly in mold machining and injection molding operations.
- b) High aspect ratio deep groove machining on the cavity plate is needed, which may require the use of very expensive machines, like: EDM (Electro Discharge Machining) machine. If you move the parting line along the z-axis, you need to machine both the cavity plate and core plate, which is also costly.
- c) High aspect ratio and little/no draft angle in vertical (or injection) direction will make it very difficult to eject the molded part.

Solution b)

Normally, the clamping capacity of injection molding machines is over 10 tons and the magnitude of it is not an important parameter with few exceptions. Mold design for better quality of parts and lower cost process is much more important than the minimum clamping force. When you choose a parting line or plane, the direction of injection is of more importance, in general.

- i) Parting plane B and C are much better choices than plane A, with a gate location marked in Figure 1 above. Because,
- ii) The injection direction normal to the parting plane B/C does not require any sliding core to form the whole feature at the bottom. Also, slight draft angle can be given to the side of cavity, with a conventional milling machine. With the gate position marked in the figure, a weld line will be formed across the hole from the gate. Since the weld line is formed while the melt temperature is still hot, quite strong weld line will be formed and remain as strong as the bulk.
- iii) Projected area on the plane B = $6 \times 10 - 3.14(1.5)^2 = 52.9 \text{ mm}^2$ (it's the same for C), Clamping force is $F = 6000 \times 52.9 / (25.4)^2 = 492 \text{ lbs} = 0.5 \text{ ton}$

Problem 3:

The following pictures show one half of an injection molded housing. With the basic design rules for injection molding in mind, is this a good design? Explain why/why not and suggest any improvements you might want to make. If you would like to, you can make a sketch to explain your thoughts.

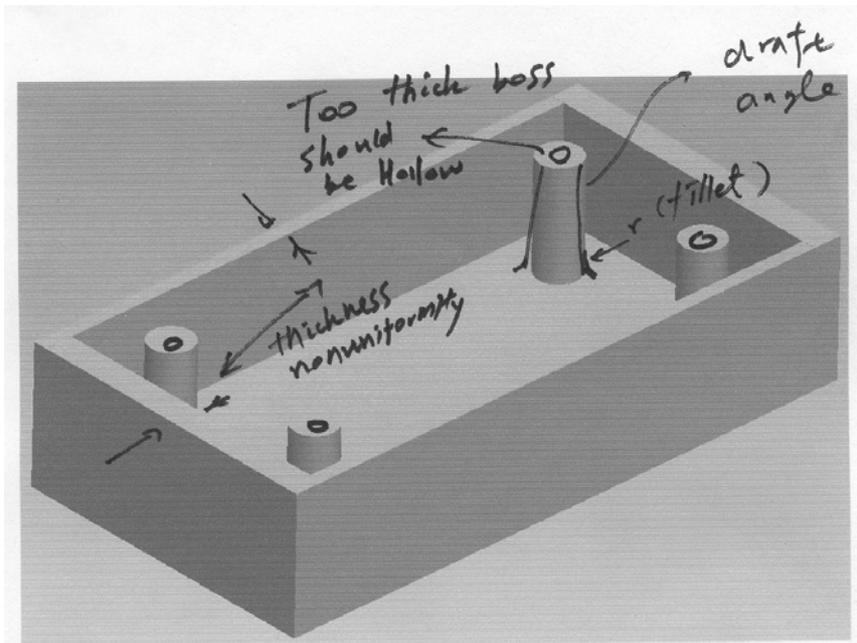


Figure 2

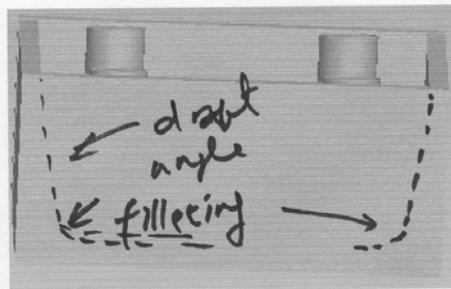


Figure 3

Solution

Injection Molding Design Guidelines

Much has been written regarding design guidelines for injection molding. Yet, the design guidelines can be summed up in just a few design rules.

- Use uniform wall thicknesses throughout the part. This will minimize sinking, warping, residual stresses, and improve mold fill and cycle times.
- Use generous radius at all corners. The inside corner radius should be a minimum of one material thickness.
- Use the least thickness compliant with the process, material, or product design requirements. Using the least wall thickness for the process ensures rapid cooling, short cycle times, and minimum shot weight. All these result in the least possible part cost.

- Design parts to facilitate easy withdrawal from the mold by providing draft (taper) in the direction of mold opening or closing.
- Use ribs or gussets to improve part stiffness in bending. This avoids the use of thick section to achieve the same, thereby saving on part weight, material costs, and cycle time costs.

For the case of figure 2/3, the points to note are the *radii* (missing!) and *thickness* (not uniform!). Also, on closer inspection a *draft angle* is missing.

a) Radii

- Sharp corners greatly increase the stress concentration. This high amount of stress concentration can often lead to failure of plastic parts.
- Sharp corners can come about in non-obvious places. Examples of this are a boss attached to a surface, or a strengthening rib. These corners need to be radiused just like all other corners. The stress concentration factor varies with radius, for a given thickness.
- In addition to reducing stresses, fillet radiuses provide streamlined flow paths for the molten plastic resulting in easier fills.

b) Thickness

- Parts should have a uniform wall thickness. 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.
- Parts should be designed with a minimum wall thickness consistent with part function and mold filling considerations. The thinner the wall the faster the part cools, and the cycle times are short, resulting in the lowest possible part costs.
- Also, thinner parts weight less, which results in smaller amounts of the plastic used per part which also results in lower part costs.
- 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 to warping or twisting. If this is severe enough, the part could even crack.

c) Draft angle

- Drafts (or taper) in a mold, facilitates part removal from the mold. The amount of draft angle depends on the depth of the part in the mold, and its required end use function.
- The draft is in the offset angle in a direction parallel to the mold opening and closing.

Problem 4:

You are going to make tools for your Yo-Yo in two weeks. You should define your functional requirements first before you seek design parameters, which meet the requirements. Please define the functional requirements of your own Yo-Yo and try to find design parameters that fulfill those requirements. Show the mapping process in-between “What” and “How” at the top two levels. This is not a single solution problem. In the world of design, you, as a designer, have all the power to explore any possible design (at least until it is proven to be a poor one). Feel free to sketch your design on a piece of paper.

Solution

This is a progressive type problem you will find your own solution during the course of 2.008 laboratory sessions. Discuss your design with your lab group members before the lab session IV and try to synthesize a better design solution for your group's Yo-Yo.

The following paragraph gives an overview of how your functional requirements, manufacturing restrictions and design parameters might look like.

Functional requirements (FR)

- FR1: playability (the yo-yo has to work in the end)
- FR2: additional requirements set up by your group (e.g. exchangeable parts, looks cool, good lab grade with nice presentation, etc.)
- FR3: pass drop test could be another FR for your Yo-Yo
- Remember that the functional requirements are the WHAT you want the yo-yo to accomplish. “Has a radio tracking device embedded in it” is a design parameter; “Is easy to find” would be the functional requirement.

Design parameters (DP)

- Top level
To meet FR1, you need to have a DP1: the device should look like a Yo-Yo (a rounded mass around the rim to transform potential energy to rotational energy, and vice versa). If you have additional FR, for example “FR2: Easy to find”, you need to find a solution to meet the FR2 such as, DP2: Embedded radio tracking device. DP3 for FR3 could be a rigid design with no thin parts sticking out.

Since you are supposed to make your product with the processes available at the shop, we need to think about the mapping process between DP domain and Process Domain.

- 2nd level
To meet DP1 in the process domain, PV1 should be moldability of your DP1. PV2 may be the possibility to integrate the radio device into your Yo-Yo. Since we don't have a detailed design yet, we cannot go further down at this process domain and need to go back to the FR domain one level down (zig-zagging).

For a given DP1 and PV1, FR1 needs to be further broken down to fulfill what you want. You may need to have a bigger rotational moment of inertia (more weight into the rim of the yo-yo (FR11), which will conserve more energy while rotating), to have a bigger radius of rotation for the string windings (FR12), a nice fit to average size hands either for lefty and righty (FR13), safety for young kids (FR14), etc. (These may differ for each designer.) FR31 might be that the Yo-Yo doesn't fall apart when dropped by a certain distance, FR 32 that it doesn't break when dropped.

DP11 may be the cross sectional shape of the Yo-Yo. DP12: Gap for string (about 1.5 of the string diameter.), DP13: Outer dimension of Yo-Yo. (look, this will be constrained by the machine and mold size, when you map this one to the process domain.) DP14: Kind of plastic material or non-sharp edges. When you map between DP's to PV's, the design for injection molding needs to be considered, such as; thinnest possible uniform thickness, etc. DP31 can be the overlapping of the different parts, DP32 the wall thickness or the inclusion of ribs.

If you have FR2 in your design, you then need to map between FR2 and DP2 by defining children of FR2 for a chosen DP2.