University of Arizona
College of Optical Sciences

Name: Nachiket Kulkarni

Course: OPTI521

Topic

Plastic Injection Molding

Submitted to

Prof. J. Burge

Date

1. Introduction

In daily life, we come across several optical technologies from barcode scanner to printers, mobile phones to complicated optical systems, from small medical devices like oximeter to endoscopes and much more. For optical technologies to develop, not only innovations are needed but also efficient ways to manufacture must be developed. For these technologies to reach out and find applications in the market, these technologies must be produced in mass and available at a low price. Injection molding is one of the methods to achieve mass production of the optical components.

Plastic is used as an alternative to glass in the manufacturing of optical components. Though they are limited by the physical characteristics of the materials and some small manufacturing tolerance capabilities, plastic elements can be very economical, especially in high volume production. Figure 1 shows some of the plastic optical components.

Figure 1: Plastic Components

2. Principles of Injection Molding

Injection molding is a manufacturing process where parts are produced by injecting material into a mold. Injection molding can be performed with a host of materials, including metals, glasses, elastomers, confections, and most commonly thermoplastic and thermosetting polymers. The material is fed into a heated barrel, and forced to a mold cavity, where it cools and hardens, hence, forming the shape same as the mold cavity.

A thorough understanding of the whole manufacturing process is essential to produce precision plastic optical components is needed as a high-quality final product is needed.

a. Injection machine

Figure 2 shows a schematic diagram of an injection molding machine.

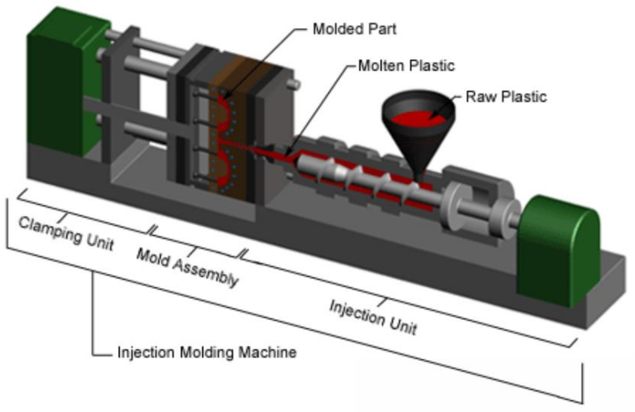

Figure 2: Injection Molding Machine

The raw material (granules) are put in the hopper and is passed through a heated a barrel where it is heated to plasticity and then forced or injected through a nozzle into a relatively cool, temperature controlled mold, where it sets up very fast. Molding cycles can be short, between 30 and 600 seconds based on the size of the component. Using multi-cavity molds, each shot can produce many pieces. After cooling the mold opens and the molded sample is ejected. The parts are removed from the gates and runners and require no further finishing processes.

b. Mold Design

b.1. Part Model

The mold is designed on CAD software, and it is our guide in mold design. Optical parameters are considered to create an optical mold.

b.1.1. Shrinkage

Plastic injection molding shrinkage is the contraction of a plastic molded part as it cools after injection. Most of the part shrinkage occurs in the mold while cooling, but a small amount of shrinkage occurs after ejection, as the part continues to cool. So the mold must accommodate shrinkage effects. Shrinkage depends on factors like molding material, cavity temperature, pressure, wall thickness, etc. Figure 3 shows an image of the shrinkage.

b.1.2. Optical/Surface equation

Some optical surface doesn't follow correct geometric equation. However, if equations are provided, it is necessary to check if shrinkage allowance has been applied. It is recommended that for a customer to have applied the shrinkage tolerance before providing it to mold maker.

b.1.3. Optical insert design

It is a crucial part of the manufacturing process. Extracting the optical insert from the rest of the mold design will give us the optical surface on a piece of steel so one can determine how to manufacture and optically polish those surfaces. Insert molding reduces assembly and labor costs, reduces the size and weight of the part, improves component reliability, and delivers improved part strength and structure with enhanced design flexibility

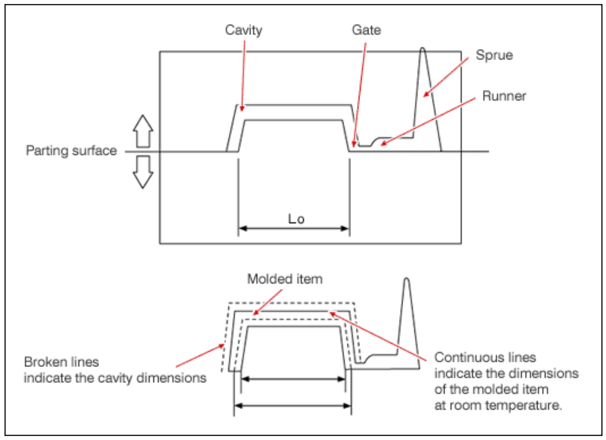

Figure 3: Shrinkage allowance

b.1.4. Split lines

A mold has two parts the cope and drag. These are separated by a split line or a parting plane. The customer and mold builder will have to choose where the split lines will be in the mold. This is used to design runner and gating system. Figure 4 shows the parting line.

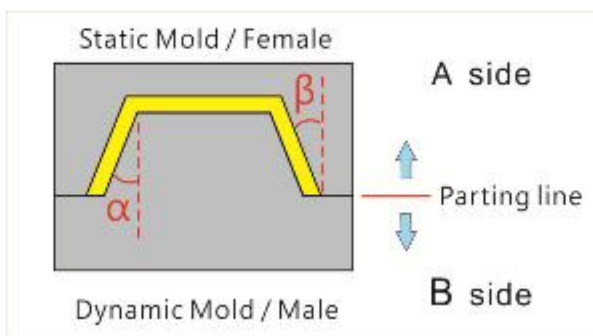

Figure 4: Parting line/split line

b.2. Mold Flow

The flow of the hot fluid in the cavity must be smooth. Mold flow analysis will prove out our part design, wall thickness cross-sections, gate size, cooling circuits, filling capabilities through the runner, and gate. This is necessary step of verification as removing material from the cavity after cooling becomes severe if any of these parameters is compensated.

b.3. Optical Inserts

Optical insert design is determined by the shape, aperture, etc. of the optic. Materials are then chosen which depends on the weight or polishability. Sometimes if a part is molded and if its thin or thick, then there must be an adjustability feature in the optical insert. Manufacturing tolerances are considered so that correct optical alignment is achieved.

b.4. Thermal Consideration

The mold requirements are determined by the temperature of operating parameters of the plastic material. For an optical mold, there are many factors made like what is average heat level, if the mold is cold or run, are cooling, and heating lines are needed, etc. The thermal properties of the plastic to be used is necessary. Measures must be taken to not increase the heat level above fire point.

b.5. Gates and runners

The gates and runners regulate the proper flow of fluid from the sprue to the cavity. The runner system is the passageway for plastic to travel from the sprue to the gate. The runner system is vital on filling cavities. If the runners are too small in size, the mold cavities will not fill correctly. If the runners are too large, then the cooling time will be increased and cycle time decreased. Proper runner design can reduce the effects of stress, sink and weld marks. Gating is critical because we do not want to put undue stress into a part. A gate permits enough flow to fill the mold cavity and compensates for shrinkage. Figure 5 shows the runner and gating system.

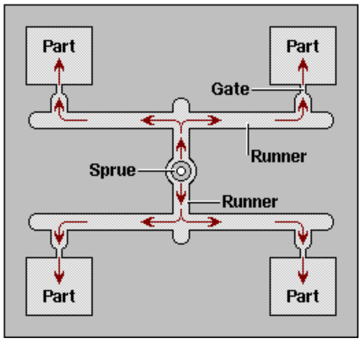

Figure 5: Gates and Runner

b.6. Ejection

The pellets of the plastic are transferred to a barrel from a hopper which is heated to the melting temperature of the polymer. The molten polymer is ejected into the mold via sprue. The type of ejection can be pin ejection, blade, air ejection, etc.; pin type being the simplest. These depend on the cross-section and size of the cavities. Sleeve type is used for cylindrical cores. Stripper plate ejection is preferred for components with larger areas. Blade ejection is used for rectangular cross sections.

b.7. Venting

When the molten liquid enters the cavity, it fills it replacing the air. So a provision must be made to allow this gas out. For a well-molded product, a proper venting is needed. Improper leads to damage of the cavity surface. Without proper venting, the more human intervention will be required with cleaning on a regular basis, and human intervention means, unfortunately, scratches or other damage. If the mold is vented properly, you will produce a better part with less pressure and less stress. Figure 6 shows the purpose of venting.

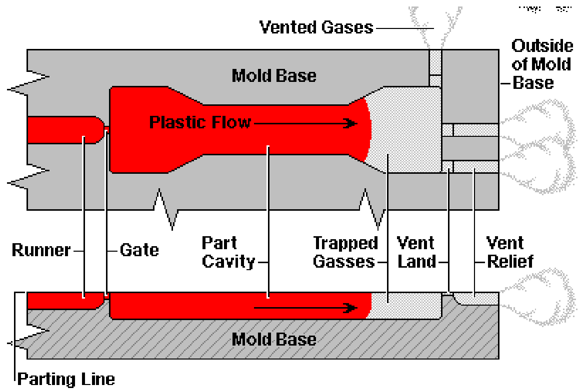

Figure 6: Venting

3. General Part Design

For a part to be molded, design is key to successful molding. Part design includes the part to be molded and mold design which is a CAD model and a control drawing. For an optical mold, optical specification and surface quality are also specified.

a. Geometric Shape

During the mold design, the product has to be considered during the design phase. As a mask fits the face form similarly, the cavity is designed such a way that final product fits completely in it. Based on the application, the best geometric shape is chosen. Assume a slab is to be fabricated using injection molding, a cuboidal cavity is made. Two rules of thumb for designing a cavity is the dimensions of the cavity must be bigger than the product to be fabricated to accommodate shrinkages. Secondly, a draft of 3-5 degrees is to be provided to vertical surfaces so that once the molten fluid cools, drawing of a part from the cavity is easy. It the two mold die have mating parts, there must be a small gap left.

b. Optical Surfaces and control parameters

Once you have chosen the particular geometry to design your product, front and back curves and their associated optical construction points, need to be defined. For a conic or an aspheric profile to be created, the construction points must be specified by the customer. A 3d design of a complex geometry is created using these. These points need to be captured and included in the part and mold design for manufacturing purposes. A mold is created using CNC. To obtain a shape of a conic, these construction points are used to program the movement of the tool.

c. CAD files

The customer should provide a 3D CAD model, a 2D control drawing, and an equation if required, along with a SAG table, which is a 2D plot of a spline that confirms the curvature of the equation and the CAD geometry. The 2D control drawing has the optical molded part requirements like optical callout, engraving, etc. The customer should also provide a full 3D model, including all aspects of the job: shrinkage factor, engraving, possible choices for gate location or areas where we cannot locate a gate, ejector locations, and any critical features.

d. Optical Correction

Most of the molds made have even wall thickness. However, in some optical molds, the even thickness of wall doesn't meet the requirement. Therefore, the thicknesses are optimized to obtain required result. Optical correction sometimes leads to manufacturing errors. Depending on the product, thick to thin cross sections need to be optimized so that they do not work against each another during the mold fill process. It is also important to make sure the part is designed to avoid having trapped gasses and voids in the mold.

e. Raytracing

It is the best way to confirm the optical performance. All optical designs are ray traced first before sending to manufacturing. Sometimes while optimizing in the optical software, results are obtained such that they are not feasible for manufacturing. The designer must consider such situations.

4. Molding Material

The customer and designer together choose molding material and manufacturer. The manufacturer suggests the material grade based on the requirements. Materials are chosen based on the requirement. The material impacts in the change of mold design. For example, if polycarbonate in used, mold must have sufficient venting. If polyurethane in used, it requires a perfect parting line. Polycarbonate needs a hot mold and polyurethane needs a cold mold. Some of the commonly used polymers are polycarbonate, polystyrene, acrylic, zeonex, polyurethane, etc.

a. Selection

The materials are selected based on optical clarity, impact resistance, transmission, haze, viscosity and refractive index.

b. Additives and coating

Material selection determines the type of coating like anti-scratch, anti-fog, mirror coat, antistatic, etc. Static charges are deposited on the surface which attracts the dust. This can be neutralized using ionized air. Antistatic agents like silica-organic liquids may be applied by dipping. These are also anti abrasive. For 98% reflection, gold coating is preferred.

5. Optical Mold Special Tooling

For a good optical mold, special features like laps, locking mechanisms, guide plate, etc. are needed and yet rarely seen by the customer. These helps in the proper optical finish and contribute to manufacturing and inspect the mold inserts, so an appropriate mold is designed.

a. Optical geometry and special tooling design

The optical geometry determines the tooling and governs what are the manufacturing processes to be used to make the mold. For example, for a sphere part will require turning, lapping, grinding and polishing. On the other hand, for an asphere needs different finishing operations. A freeform needs hand polishing. The design of the special tooling needs to be done at the same time as the mold design, as we discussed: the fixtures to hold the inserts during manufacture, the polish holders required for any holding operations, fixtures that might be required for inspection, laps needed for any lapping and polishing processes, or diamond turning holders.

b. Surface Finish

The finish requirement of the optic determines the method by which the mold inserts are to be finished. For simple geometries, abrasive lapping and polishing would be sufficient. However, for aspheric surfaces, sophisticated techniques like diamond might be needed.

c. Alignment

Aligning of the two mold halves, cope and drag and optical insert with respect to these halves is necessary. The two halves must be aligned and to retain this while injecting fluid, a locking mechanism is made. Inserts need to be installed so that optical axis is brought into line.

6. Inspection

a. Steel vs Plastic

The first step is to determine if the steel is at place. Each component like inserts, surface finishes and parts of mold which contribute in producing plastic parts are inspected. If everything turns correct, it is assumed that a proper molded product is produced. The next step is inspection of the part. Optical testing criteria is resolution, power, scratch/dig, etc.

b. Testing

Optical testing methods can be contact measurements such as coordinate measuring machine (CMM) or non-contact like laser scan. An optical telescope is used to look for optical clarity and lines of resolution. Polarizers can be used to analyze stress. An interferometer can measure the form and figure.

7. References

- http://info.crescentind.com/blog/bid/69598/What-is-Insert-Molding-for-Plastic-Components
- http://www.misumi-techcentral.com/tt/en/mold/2009/07/0001-what-is-the-molding-shrinkage-phenomenon.html
- http://www.dmeuniversity.net/frmLessons.aspx?CourseID=4&LessonID=53
- https://en.wikipedia.org/wiki/Injection_mold_construction#Ejection_system_types
- David Borque, “Manufacturing Plastic Injection Mold”, Novel Optical Systems Design and Optimization XI, 706117(2008)
- Ulrich Greis, G. Kirchhof, Agfa-Gevaert AG, “Injection Molding of Plastic Optics”, Optical Surface Technology, 0381(1983)