Lecture 4.5: Injection Molding

Injection molding is the one of the most commonly used manufacturing process for the plastic components. It is used to manufacture thin walled plastic parts for a wide variety of shapes and sizes. In this process, the plastic material is melted in the injection chamber and then injected into the mold, where it cools and finally the finished plastic part is ejected.

Working Principle

In this process, the plastic materials usually in the form of powder or pellets are fed from hopper into the injection chamber. The "piston and cylinder" arrangement is used to forward the material inserted from the hopper in to the injection chamber. The plastic material is heated in the injection chamber with the application of heating elements. The cooling system is also used to maintain the temperature of the injection chamber. The molten plastic material is then injected into the mold cavity through a nozzle. The molded part is cooled quickly in the mold. Thereafter, the final plastic part is removed from the mold cavity. The process cycle for injection molding is very short, typically between 2 to 60 seconds. The complete injection molding process is divided into four stages: clamping, injection, cooling and ejection.

Clamping: The two halves of the mold must be tightly closed, before the molten plastic material is injected into the mold. One half of the mold is attached to the injection unit (nozzle) and other half is allowed to slide on the guide ways. The clamping of mold is operated hydraulically which it pushes the moving half part of the mold towards the fixed part to make an air tight chamber. The force and the time required to close and open the mold depends upon the machine capability.

Injection: During this process, the plastic material is melted by the application of heat and forwarded through the piston towards the nozzle and finally into the mold. The function of torpedo in the heating zone is to spread the molten plastic into the thin film. The molten plastic is then injected into the mold cavity quickly. The amount of material that is injected into the mold is referred to as the shot volume. The injection time can be estimated by the shot volume, injection power and pressure. The schematic of injection molding process is shown in figure 1.

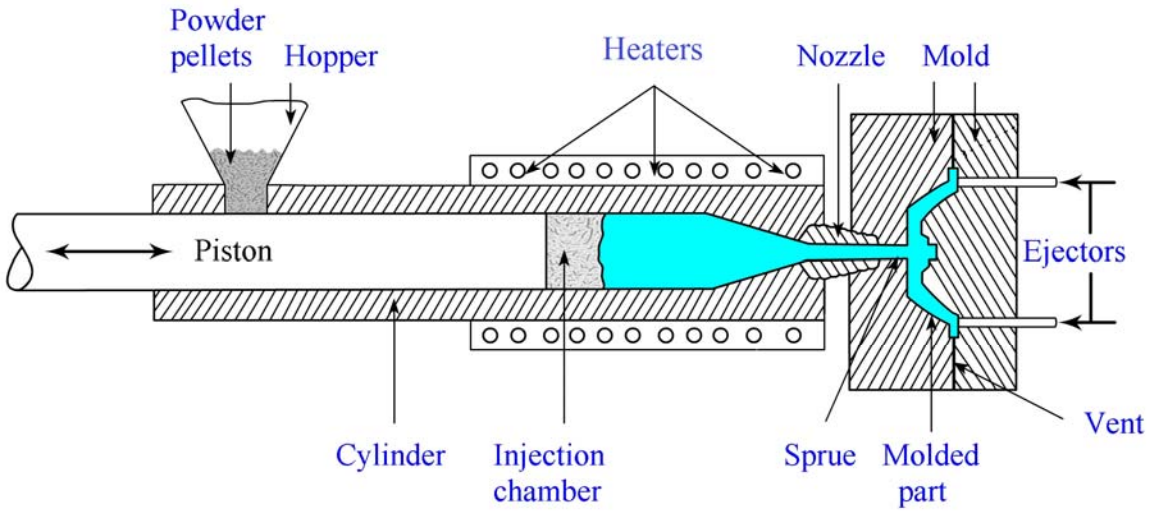


Figure 1 Injection molding setup

Cooling: The injected molten plastic begins to cool as soon as it comes in contact with the mold surfaces. As the molded part cools, it will solidify into the desired shape of the product.

Ejection: The molded part, which is attached to the rear half of the mold has to be ejected from the mold. When the mold is opened, an ejector mechanism is used to push the part out of the mold. Force must be applied to eject the plastic part because during cooling the molded part shrinks and adheres to the mold surface. A mold release agent should be sprayed onto the mold surfaces prior to injection of the material. Once the molded part is ejected, the mold will be closed for the next shot to be injected.

The important process parameters that have to be considered during the injection molding process are: injection temperature and pressure, shot volume, mold temperature, cooling time, ejection temperature, and cycle time.

Some of the common injection molding defects are flash, blister, warping, bubbles, unfilled sections, jetting, sink and ejector marks. These defects can be eliminated by optimal selection of the process parameters.

Materials Used

The injection molding process can be used to process materials such as Acetal, Acrylic, Acrylonitrile Butadiene Styrene (ABS), Cellulose Acetate, Polyamide (Nylon), Polycarbonate,

Polyester, Polyether Sulphone (PS), Polyetheretherketone (PEEK), Polyetherimide, Polyethylene, Polyphenylene Oxide, Polyphenylene Sulphide (PPS), Polypropylene (PP), Polyvinyl Chloride (PVC), and Elastomers.

Applications

This process can be used to manufacture thin walled plastic housing products which require many ribs and bosses on the interior surfaces. These housings are used in a variety of products including household appliances, electronics, and automotive dashboards. Other common thin walled products include different types of open containers, such as buckets. It is also used to produce several daily use items such as toothbrushes or small plastic toys, many medical devices, including valves and syringes.

The advantages and disadvantages of the injection molding process are given below:

Advantages

- Higher production rate
- Close tolerances on small intricate parts
- Minimum wastage of material
- Complex geometry can be easily produced

Disadvantages

- Tooling cost higher
- High setup cost
- Large undercuts can’t be formed