Construction principles of injection molds

D. Meyer

INTRODUCTION
The mold is the 'shaping element' in the process of injection molding with thermoplastics. The construction of the mold has a significant influence on the total molding process and furthermore affects the product quality, the production cycle and the possibility for automation as well [1].

The basic steps for the design of an injection mold are listed in Table 1.

Important for the optimal design of molds is a proper knowledge of the use, assembly and the service conditions of the injection molded items. From the viewpoint of the polymer selection, it is necessary to know the thermal, mechanical, electrical and chemical requirements for the molded part. There is a number of specific design aspects relating to individual tasks; some of them are listed below.

• Automatic ejection (or demolding). The molds are equipped with mechanical or electric driven screw-thread cores for moving parts of the mold.
• High-speed-molds should preferably have a design with a take-out process using horizontal handling systems via the shortest route. In the vertical process, the pieces usually fall down onto conveyor belts (that can be directed by a vertical directed breeze). A special design is called the 'non-open mold concept'. In principle, this concept does not
require a demolding unit. Article demolding takes place as a parallel operation to ‘mold opening’, which only requires a very short stroke.

- Placing inserts, in-mold decoration. The mold construction covers also the handling of sheets (decorations, labels, smartcards, etc.), drapery insert (auto-panels, furniture, etc.).
- Production of hollow parts. Gas-assisted injection molding [2] is used to produce hollow bodies with smooth surfaces (auto mirrors, handles).
- Multi-color or multi-component molding. For multi-color molding, two or three colors of the same material are used whereas, for multi-component molding two or three various polymers or the same polymer with different melt flow index (MFI) values are used.

MOLD DESIGN

The mold design starts with deciding the main dimensions (Table 1). Rectangular molds are usually preferred for a better adjustment and fixing at the platen area than round molds. The rectangular molds allow also an easier automatic tool-change. The selection of the cavities is a significant decision that should be based on the amount and price of the product. Three-plate molds (Figure 1) produce twice as much in the same time as two-plate molds at the same clamping force of the injection molding machine. There is also a wide variety of sprues that can be used: stick, point, film, cold and hot runner sprue.

Ejector pins (Figure 1) are responsible for ejecting the molded pieces. They are usually located on the opposite side to the sprue. Round, profile