Pruner, Nesch

Understanding Injection Molds

Harry Pruner
Wolfgang Nesch

Understanding Injection Molds

HANSER
Hanser Publishers, Munich
Hanser Publications, Cincinnati

The Authors:
Harry Pruner,
Pruner Marketing Services GmbH, Meisenweg 7, 79341 Kenzingen, Germany

Wolfgang Nesch,
Waldstraße 81, 77933 Lahr, Germany

Distributed in North and South America by:
Hanser Publications
6915 Valley Avenue, Cincinnati, Ohio 45244-3029, USA
Fax: (513) 527-8801
Phone: (513) 527-8977
www.hanserpublications.com

Distributed in all other countries by
Carl Hanser Verlag
Postfach 86 04 20, 81631 München, Germany
Fax: +49 (89) 98 48 09
www.hanser-fachbuch.de

Library of Congress Cataloging-in-Publication Data
Pruner, Harry, author.
[Spritzgiesswerkzeuge kompakt. English]
Understanding injection molds / Harry Pruner, Wolfgang Nesch.
pages cm
Includes bibliographical references and index.
ISBN 978-1-56990-527-2 (hardcover) -- ISBN 978-1-56990-535-7 (e-book) 1. Molding (Chemical technology) I. Nesch, Wolfgang, author. II. Title.
TP156.M65P7813 2013
668.4'12--dc23
2013025198

Bibliografische Information Der Deutschen Bibliothek
Die Deutsche Bibliothek verzeichnet diese Publikation in der Deutschen Nationalbibliografie;
detaillierte bibliografische Daten sind im Internet über <http://dnb.d-nb.de> abrufbar.

ISBN 978-1-56990-527-2
E-Book ISBN 978-1-56990-535-7

Production Management: Steffen Jörg
Coverconcept: Marc Müller-Bremer, www.rebranding.de, München
Coverdesign: Stephan Rönigk
Translation: Olivia Brand
Typesetted, printed and bound by Kösel, Krugzell
Printed in Germany

How to Use the Book

The basis of the descriptions in this book are the thermoplastic molds. Divergent processes for thermoset or elastomer molds are explained at the end of the respective chapter.

In the subsequent table of contents, colored squares indicate whether the areas of thermoset and elastomers are identical, not identical, or not available at all for the thermoplastics.These markings are also continued in each section of the text, where the upper square represents the elastomers and the lower square represents the thermosets.

In the coloring of the principle figure, the following recurring colors were used:

red = Injection molded part
yellow = Second component in multi-component parts
orange = Explained component of one chapter

■ Elastomer or thermoset molds are identical
■ Elastomer or thermoset molds are not identical and are further described
■ Topic area for elastomer or thermoset molds is not available

Table of Contents

Preface

This textbook is specifically addressed for a beginner. The content provides the reader with a comprehensive and concise description of all the relevant components of an injection mold in a practical, easy to understand presentation. The chapters are designed so that they provide a complete basic knowledge of injection molds in chronological order as well as daily guidance and advice. The target group is not the designer, but the newcomers and processors in injection molding who can get a quick and comprehensive explanation of the variety of injection molds. In the foreground of the description are thermoplastic molds. In particular, the procedural aspects are highlighted in a compact form when explaining molds. Divergent processes for thermoset or elastomer molds are also described at the end of each chapter. Particular emphasis is placed on a clear didactic structure of the book, so that the readers can capture all the essentials quickly. Deeper knowledge, as designers and professionals would require in production, can be found in other publications.

We are grateful for information from the users about optimizations of this textbook. At this point we would like to give a special thanks to the companies that supported us by providing expert information, constructive criticism, and pictures, as well as all employees of the Carl Hanser Publisher who were involved in the creation of this book.

Harry Pruner
Wolfgang Nesch

1 Basic Mold Design

1.1 Assemblies of an Injection Mold

The most important elements of an injection mold, along with the common technical terms, are given in the introduction to provide a basic understanding of the technology. An injection mold generally consists of two mold halves: a nozzle side and an ejector side. The cavity inserts, the sprue systems, the cores, the ejector elements, and the cooling system are located in the mold halves.

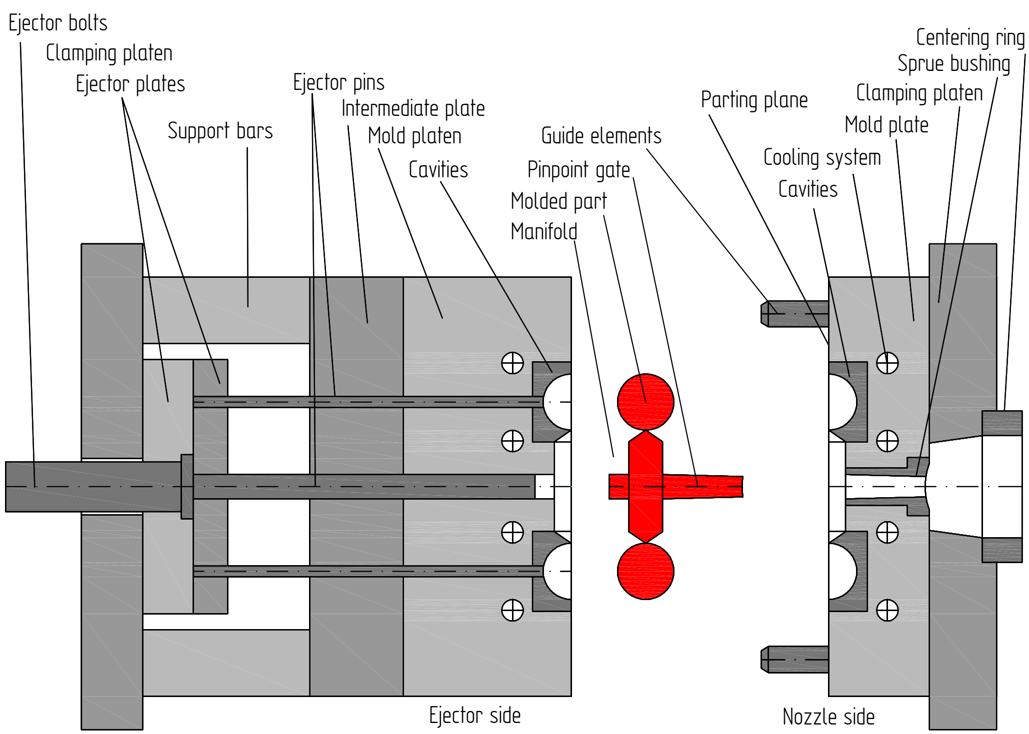

Figure 1.1 Principle of a two-platen mold

1.1.1 Phases of Design

It is important to start with a basic sketch of the mold to see which mold technology can be used. It is necessary to determine whether special functions are needed and if this concept is suitable for a fully automated production. Then the number of cavities should be evaluated, because this will affect the injection molding machine size and the dimensions of the mold. The next step is the determination of the plastic material. Is the plastic material easy-flowing, viscous, or reinforced? Does the mold need to be cooled or heated? Which sprue system is to be used?

With this knowledge, the design can begin. First, a parts drawing in 3D has to be made. Then a decision is made whether the filling simulation (Moldflow® analysis) is carried out for this part module. For a part that is exposed to mechanical stresses, a finite element analysis (FEA) and a simulation process may also be necessary.

The big advantage of this approach is that, except for the production of the prototypes, all functions are possible with computer-aided design (CAD) programs. Once this phase is completed, discussions with the customers for the detailed clarification of all points are recommended. This is followed by the release of the design. The mold and the individual components are then produced.

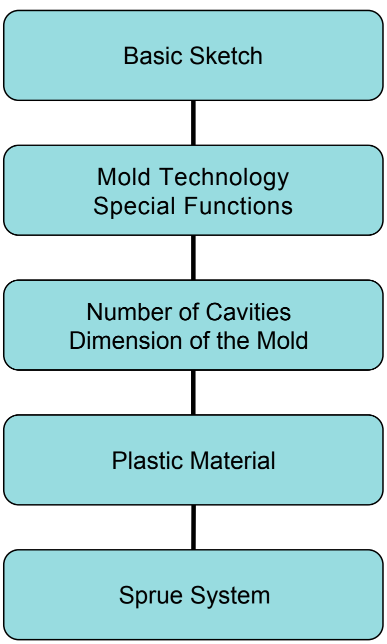

Table 1.1 Information Phase

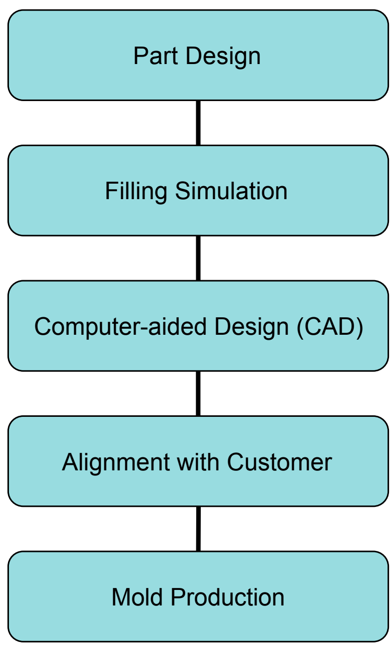

Table 1.2 Design Phase

1.1.2 Stability of Thermosets Molds

The cores and inserts should, if possible, not be assembled for thermoset molds. For the stability of the mold, it is better if they are made from one piece. For stability reasons, it is recommended for thermoset molds that one choose the master molds to be generally one size larger than for thermoplastic molds. This prevents deformation of the side walls of the mold by the high injection pressures (up to 2500 bar).

1.1.3 Wear in Thermoset Molds

Molds for thermoset processing tend to higher wear than thermoplastic molds. This is mainly due to the aggregates, such as glass fibers and mineral powder. These aggregates are abrasive. Higher requirements are also placed on the stability of the molds.

Already in the design it will be necessary to ensure that sharp edges and deflections are designed to be exchangeable. To minimize wear, sprue components made from sintered metal are used.

The mold plates and shaping components, such as cores, dies, sliders, etc., must be hardened and hard chrome plated if necessary.

It is recommended to chemically nickel plate all components that are not in the area of the parting plane because the gases and water vapor, which enter the area of the ejector plate through the ejector, are aggressive and cause corrosion.

For stability reasons, it is recommended that thermoset molds generally be one size larger than needed for thermoplastic molds. The mold plates of the thermoset molds and the shaping components such as cores, dies and sliders should be through-hardened.

1.2 Nozzle Side

Simple injection molds consist of two halves, the nozzle and the ejector side, which are both built from multiple platens. The nozzle side is the mold half that does not move during production.

In most cases, the half-shells of the cavities, also called mold inserts or mold cavities, are incorporated into the nozzle-sided mold plate of the two-plate molds. The components that belong to the sprue system, such as the sprue bushing (usually in combination with a cold runner manifold system), are also located on the nozzle-sided mold plate.

The hot runner manifold with hot runner nozzles for the production of the injection molded parts is another assembly of the nozzle-sided mold plate.

The mold is mounted on the machine clamping platen and is equipped with a replaceable centering ring. Through the centering ring, the nozzle tip dips into the mold and presses itself against the sprue bushing.

For molds with higher mold temperatures, an insulation platen is attached before the clamping platen to prevent heat transfer to the machine-sided clamping platen.

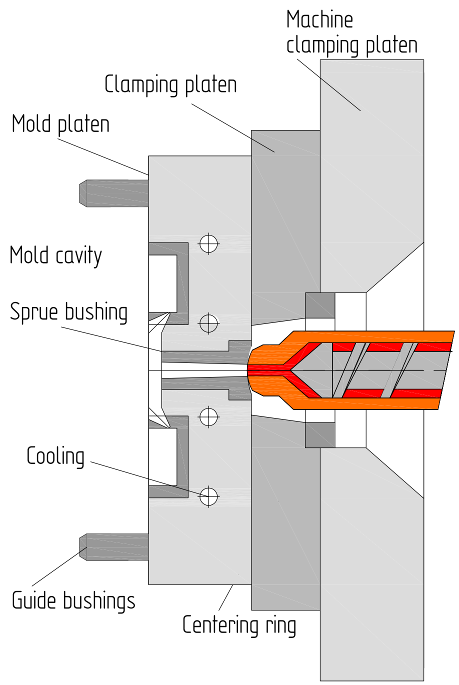

Figure 1.2 Nozzle-sided mold half

The nozzle side of the mold is mounted to the fixed machine platen. The term derives from the fact that this part of the mold is located in front of the machine nozzle.

1.2.1 Sprue Bushing

The exact connection of the machine nozzle with the mold is done via the sprue bushing. The bushing must be adapted to the machine nozzle used.

The sprue nozzle must be selected according to the different types of nozzles (e.g., cone, radius, flat, and diving nozzles). An exact coupling is of great importance. If the coupling is not tight from shot to shot, more and more mass is pushed through the nozzle. This leads to contamination and also changes the pressure transfer into the mold.

Diving nozzles are ideal for hot runner molds. The sprue bushing in these molds is heated.

Except for diving nozzles, the diameter of the nozzle mouth almost always has to be somewhat smaller than the diameter of the sprue bushing so that no undercuts are created. Undercuts can make demolding of the sprues more difficult.

The nozzle of the plasticizing cylinder during the injection process exerts a large force (nozzle contact force) on the sprue bushing. The mold is therefore subjected to very high loads at the sprue bushing, which can wear relatively quickly. Sprue bushings are therefore made from hardened steel.

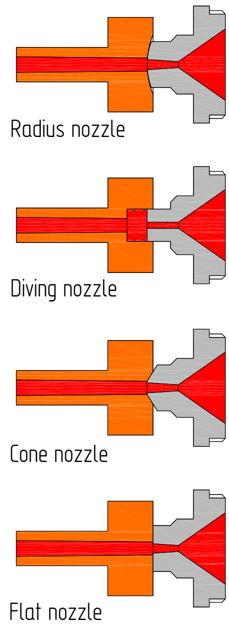

Figure 1.3 Principle of a sprue bushing

The sprue slug is incorporated into the sprue bushing. The slug transfers the plastic molding compound from the plasticizing cylinder. Large forces act on the sprue bushing during injection. Therefore, the bushing is made of hardened steel.

1.2.2 Decompression

Especially when using hot runner systems, a proper decompression is extremely important so that no plastic can escape out of the hot runner nozzle when the mold is open.

Screw Retraction

The pressurized material in the hot runner is relieved by selective retraction of the screw during the screw retraction. This prevents material from leaking out of the hot runner nozzle when the mold is open.

Diving Nozzle

A decompression alternative is a diving nozzle, which is always used when the injection machine has no screw retraction. The injection unit with nozzle head may be withdrawn during the production only to the extent that the hot channel is relieved. If the nozzle head constantly leaves the diving nozzle, air comes into the hot runner system. During injection, the air compresses and burns the plastic material (diesel effect). The consequence of this is that the injected parts have streaks and surface defects.

Diving nozzles are very common in mold making. Therefore, they are offered as standard parts. Often, a diving nozzle with a strainer insert is built in. This improves the homogeneity of the plastic mass and prevents nozzle seal by foreign bodies.

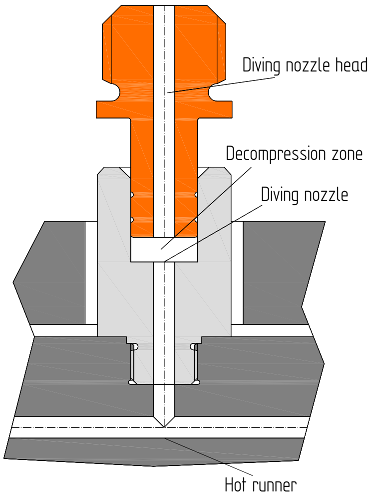

Figure 1.4 Diving nozzle system

Screw retraction or diving nozzles prevent liquid plastic from escaping out of the hot runner nozzle where it then cool and causes problems such as a material plug or threads.

1.2.3 Strainer Nozzles

Strainer nozzles are used to process recycled material, filter out foreign substances, and better mix the color pigments. They may be incorporated into the hot runner, the mold, or the machine nozzle.

When milling plastics, foreign materials can often get into the mill. Ferrous and nonferrous metal parts can be separated relatively easily because they are detectable. But wood, glass, paper, and other unfiltered foreign substances can get into the nozzle bore holes of the hot and insulating channels and clog them.

These foreign substances can be filtered out with strainer nozzles. The replacement of a clogged strainer nozzle is easy. Contamination of a hot runner however requires disassembly of the mold.

Strainer nozzles are also suitable for the mixing of color pigments. If strainer nozzles are used in the processing of colored material, less masterbatch is needed due to better mixing and distribution of the color. The material is much more homogeneous after it has passed through the strainer nozzle.

Installation of the strainer nozzle is possible both in the mold as well as in the machine nozzle.

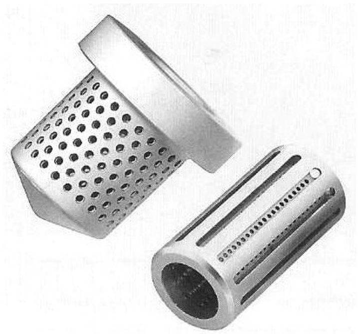

Figure 1.5 Strainer nozzles (Source: HASCO)

Strainer nozzles prevent contamination in recycled material from getting into the hot or isolation channels.

1.2.4 Nozzle Side of Thermoset Materials

Injection molds that are used for processing thermosets do not have connections for temperature control because the molds are electrically heated.

The distribution channels, if any, are more rounded. There are hot runner systems that are not electrically heated but tempered with water. In some molds, the mold cavities are directly incorporated into the nozzle-sided mold plate.

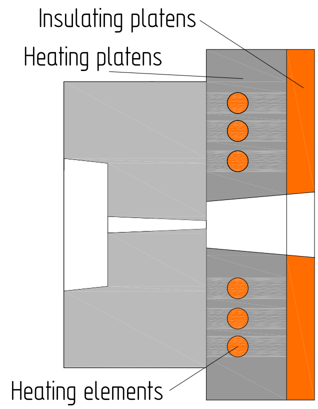

Figure 1.6 Nozzle side of thermoset molds with hot plates

> Thermoset molds must be designed to be more stable because of the increased stresses. They are therefore through-hardened.

1.2.5 Nozzle Side of Elastomer Molds

Not all materials used in the elastomer injection molding process are free-flowing, so different metering systems are used. The free-flowing masses are fed to the feed zone of the screw through a funnel. Some elastomers are supplied as a band, and others are pasty. They are pressed from cartridges to the screw. The two-component liquid silicone is supplied to the screw from two different containers using a mixer.

The nozzle side of elastomer molds with hot plates corresponds to the nozzle side of thermoset molds with hot plates (see Figure 1.6).

> Injection molding is today the most widely used technique in elastomer processing. In contrast to the thermoplastic molds, which are mainly cooled, elastomer molds have to be heated.

1.3 Ejector Side

The shaping cores, inserts (also called cavities), and the ejector elements are installed in the ejector side. The molded part generally remains on the ejector side when opening the mold. Depending on the level of difficulty of the demolding, different ejector devices are used.

After first examining the molded part, the type of demolding needed is determined. The following versions are available:

- Parts without undercuts can be stripped, removed or demolded with ejector pins.
- Parts with undercuts can be demolded with sliders or jaws.
- Parts with internal or external threads can be forcibly demolded, unscrewed with rotating cores, or demolded with collapsible cores.

Both mold halves each have a separately controllable temperature control system to ensure an optimum temperature control. The mold temperature is dependent on the plastic to be processed.

All injection molds can be divided into three basic modules and their respective subgroups. These are the types of demolding, the sprue technology, and the temperature control. The fourth criterion is the selection of the special design.

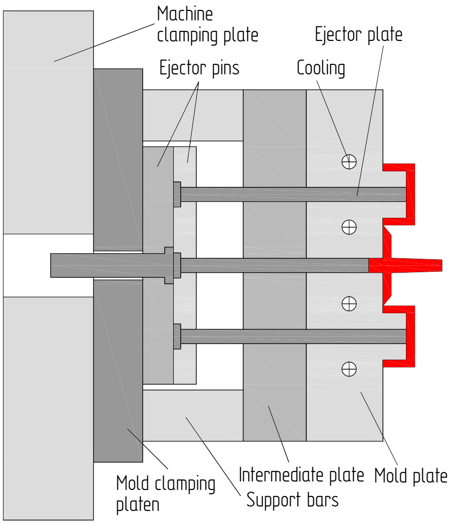

Figure 1.7 Ejector-sided mold half

The plastic part, the so-called injection molded part, largely determines the overall design of the mold. To distinguish the various applications, the molds are divided according to the demolding principle, the sprue system, the temperature control, and the special design.

1.3.1 Ejector Device

A series of standard parts is available for the many ways to eject injection molded parts. Molds with special features, however, mostly have an ejector system, which is tailored to a specific application.

In standard molds, which eject parts on the ejector side without special function, the ejector device (ejector set) is directly coupled to the machine ejector. Ejector systems are either mechanically activated by the machine directly or are hydraulically or pneumatically activated by the mold.

There are several practices to eject molded parts.

1.3.1.1 Ejector Set with Ejector Pins

The most common type of demolding is demolding with ejector pins. Ejector pins are inexpensive, standardized according to DIN 1530, and easy to install. When installing, it is important to note that the pins and bushings are free outside the guide length, so that they can be centered at different temperatures of the mold plates in the bore holes and no unnecessary frictional resistance occurs. See Figure 1.8.

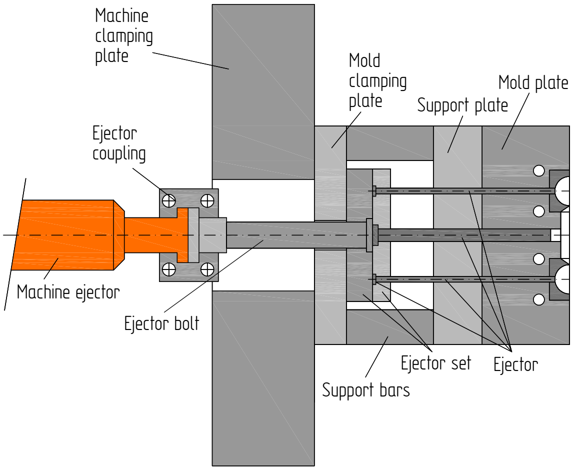

Figure 1.8 Ejecting with ejector pins

1.3.1.2 Stripper Plate

Stripper plates have the advantage that the demolding force is introduced over a large area (i.e., over the entire edge of the molded part). See Figure 1.9.

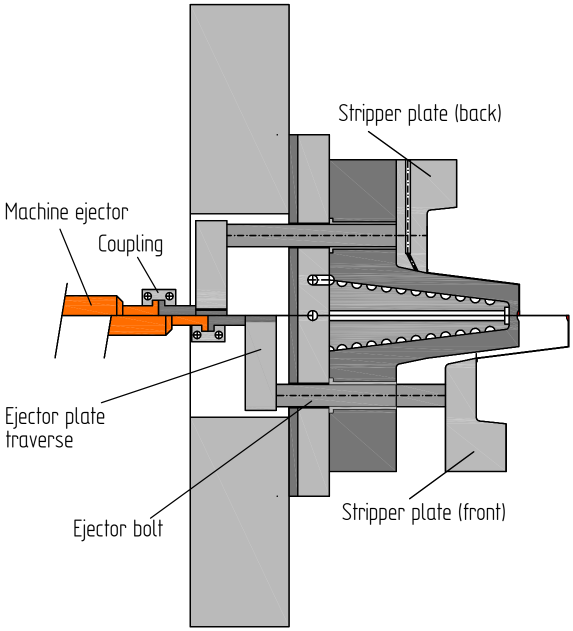

Figure 1.9 Ejecting with stripper plate

1.3.1.3 Mushroom Ejector

Mushroom ejectors are often used in the manufacture of thin-walled packaging containers such as buckets, mugs, bowls, and yogurt cups. The mushroom ejector has the advantage that it accesses a large area at the bottom of the molded part. The molded part is only raised with the mushroom ejector and then completely blown off the core using air. The mushroom ejector always accesses centrally on the molded part. Its stroke is only about 10 – 15 mm.

In single-cavity molds, the ejector mushroom is located in the middle of the mold; in multi-cavity molds, it is located on an ejector plate. The mushroom ejectors are not coupled with the machine ejector, as is common for other molds, but pushed back to the resting position by spring force after ejecting.

The mushroom ejector in the top plate and in the shaft region must be intensively cooled because packaging parts are fast moving.

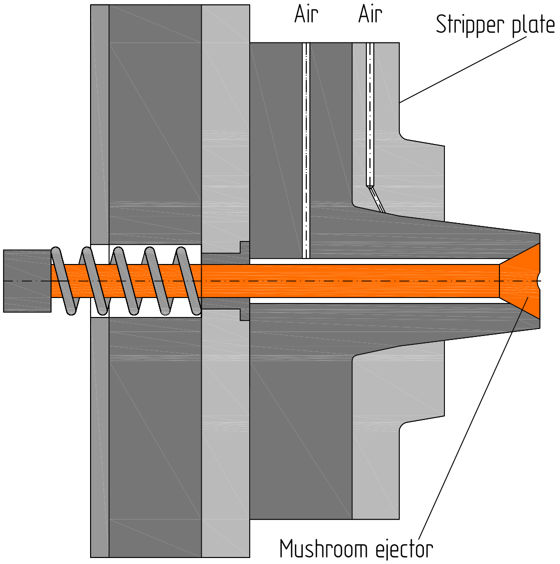

Figure 1.10 Principle mushroom ejector

1.3.1.4 Air Ejector

With the air ejector, nozzles and slots are attached at suitable points, supplied with compressed air, and controlled by control valves. The compressed air must be oil-free and clean to avoid contamination of the injection molded parts. Air ejectors are mainly used in molds for thin-walled parts.

The geometry of the parts, but also the size of the molded part determines the ejector system. Ejector pins, stripper plates, mushroom ejectors, or air valves may be the right solution. Hydraulic machine ejectors are today mainly used for actuating the ejectors. They have the advantage that time, force, and speed can be freely selected.

1.3.1.5 Ejector System for Thermoset Molds

Releasing and ejecting the molded parts in thermoset processing is done similarly to that in thermoplastic molds.

Gases and water vapor, which also contain monomer components, are drained from the mold cavity via the ejector pins. There is a risk that these monomer residues can stick in the bore hole and clog it. Recesses in the ejector prevent this. Material residues then settle in the recesses. These recesses cause a cleaning of the ejector bore hole with a repeated movement of the ejector. The venting remains in effect.

The diameter of the ejector elements should be chosen as large as possible so that a good venting is taking place. This prevents the ejector from deforming or puncturing the molded part. Except the ejector pins, there is a number of ejector aids as for the thermoplastics. These include, for example, stripper plates, wiping strips, and mushroom ejectors.

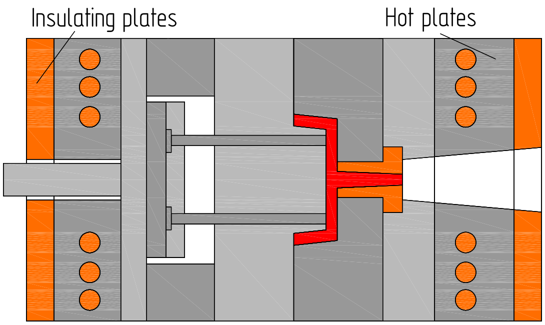

Figure 1.11 Injection mold with hot plates

1.3.1.6 Ejector System for Elastomer Molds

Molded parts made from elastomer or silicone are not easy to eject due to their fragile nature. Therefore, the ejector elements are designed larger than for thermoplastics in order to lift the parts out of their mold cavities or strip them from the cores. Often, it is nearly impossible to eject the part undamaged due to the rough surfaces present. In this case, the molded part is back ventilated. Closed bodies can be back ventilated to the point that they slide off the core almost by themselves due to their elasticity.

Molds for silicon processing in the area of the ejector plates are closed so that a vacuum builds up. The shaping lots of silicone molds are usually provided with a slight erosion or etching structure or are sandblasted. This is because silicone parts bond extremely well to polished surfaces.

There are molded parts with deep undercuts that can be demolded without sliders. The most common parts of this type are bellows. So that the bellow can be inflated for demolding, it must be closed on one side.

The ejector side of molds for elastomer and silicone processing only differs minimally from thermoplastic molds. Generally, the connections for the liquid temperature control are missing. Instead, the ejector side is heated. An insulation plate is mounted between the mold and the machine clamping plate for the thermal insulation.

1.3.2 Draft Angles

When demolding, the molded part is first released from the surface of the mold by compressed air. The draft angles support a quick and easy removal.

To demold injection-molded parts easier, all interior and exterior surfaces should have an inclination in the demolding direction. This inclination is called the draft angle. In containers, boxes, or similar components, a draft angle of 0.5° to 3° is usually easy to achieve. Smaller draft angles are possible but require a slower removal.

Some components do not permit a draft angle for functional reasons. In this case, the respective sector must be released using sliders. If it is not possible to install a slider, though in any case, the surface has to be hard chrome plated. On the chrome layer, the sliding properties are much better than that of steel.

Besides missing draft angles, shrinkage, and insufficient or inappropriate release agents, not-quite-cured parts and molds that are not cleaned are typical errors that prevent smooth demolding.

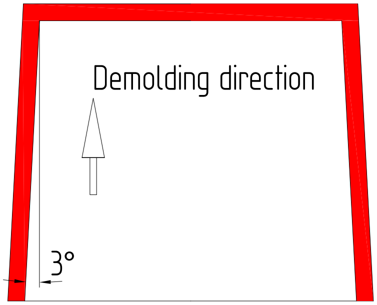

Figure 1.12 Principle of draft angles

The larger the draft angle, the faster demolding can be done. The draft angle should not fall below 0.5°.

1.3.3 Ejector Coupling

Today, in most cases, ejector couplings are used for the withdrawal of the ejector plate. The ejector coupling provides for a form-fit connection of the hydraulic machine ejectors with the molds.

Ejector coupling systems range from simple plug-in to quick-clamping systems for automatic coupling and de-coupling. With the simple plug-in systems, the connection is usually carried out with a cross bolt or a screw. There are also ball locking systems that are secured with a sliding sleeve.

If the mold is connected to the injection molding machine with a quick-clamping system, the ejector coupling must be automatically coupled. This operation can be performed pneumatically, electrically, or hydraulically. The ejector coupling is firmly connected to the machine ejector through the threaded stem. For the pneumatically operated ejector couplings of the most common version, the coupling is actuated with a pressure of 6 bars. Subsequently, the coupling bolt is locked in a mechanical way without air-applied pressure.

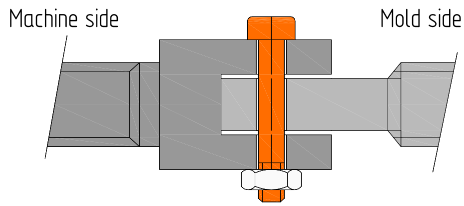

Figure 1.13 Ejector quick-snap coupling (for small molds)

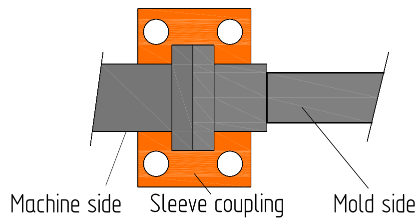

Figure 1.14 Principle of ejector sleeve coupling (for large molds)

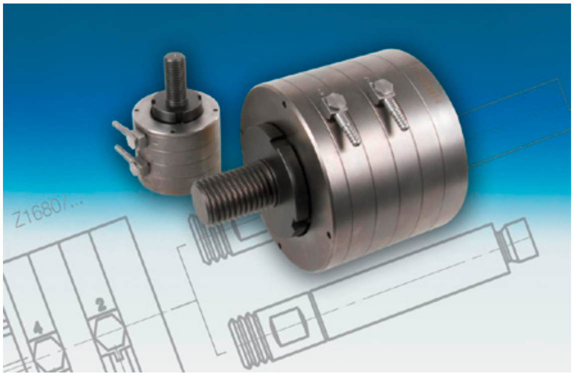

Figure 1.15 Pneumatic ejector coupling (Source: HASCO)

The ejector coupling provides a significant reduction in setup times for frequent mold changes.

1.4 Buoyancy Forces in the Mold

If the plastic melt fills the cavity with the injection pressure necessary for the injection process, an injection pressure of 250 to 2000 bar can be achieved, depending on the material. A buoyancy force acts over the projected area of the molded part.

If the buoyancy force is greater than the clamping force, plastic material is pressed into the parting plane. This is referred to as over-molding, and a plastic burr is created at the molded part. When this occurs, the locking or clamping force of the injection molding machine is too small. The clamping force of the machine should be at least 10 % higher than the buoyancy force in the mold.

The specific injection pressure results from the ratio of flow path and wall thickness in combination with material coefficient values such as viscosity and additives such as glass fibers, talc, color additions, and so on.

The flow path, measured from the injection point, is the path that the plastic has to travel in the injection molded part, including all diversions to the farthest point in the molded part. The wall thickness is the value, which prevails in the majority. An example: a flow path of 120 mm and wall thickness of 0.8 mm results in a ratio of 150 : 1.

The cavity pressure diagram shows that the expected cavity pressure (P_{WI}) is at approximately 450 kN/cm^2. For example, if the projected area (A) of the molded part is 200 cm^2 in size, the required clamping force is 900 kN. With a safety margin of 10 %, a machine with 1000 kN clamping force is required. This method of calculating the clamping force can be applied to all injection molds.

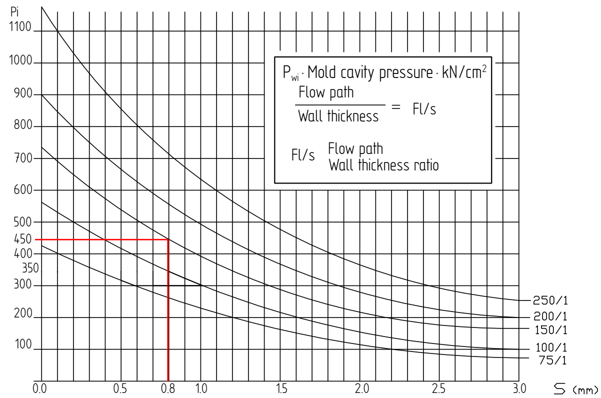

Figure 1.16 Relationship between cavity pressure and wall thickness

The mold buoyancy force (F_A in kN) is the product of the mold cavity pressure (P_{WI} in kN/cm^2) multiplied by a projected area in the parting plane of the molded part (A in cm^2). If the buoyancy force is too high, the molded part is over-molded. To ensure that the mold does not open through the buoyancy force, the holding force (= clamping force) has to be greater than the buoyancy force.

$$F_A = \frac{P_{WI} \cdot A}{100}$$

1.5 Mold Protection

To secure the mold against stuck parts during production, all injection molding machines are equipped with a mold protection, which reduces the closing pressure and the speed over an adjustable distance during the closing process.

This process protects the mold but requires an additional time of 1 – 2 seconds for the opening of the mold after the disturbance. Therefore, safety gate valves for the mold are better, because they do not allow a closing movement of the machine when a part gets stuck in the mold.

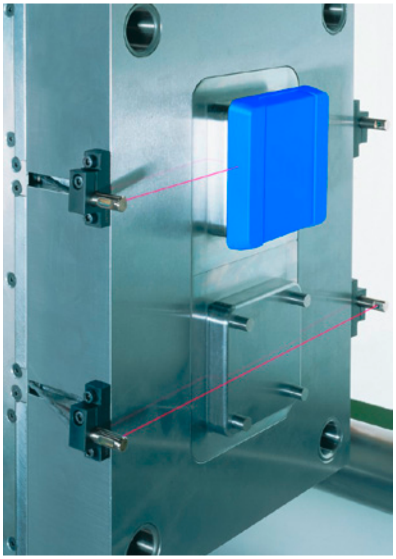

Figure 1.17 Infrared mold protection (Source: HASCO)

1.5.1 Light Barrier/Failure Scale

When using single-cavity molds, a light barrier or a failure scale, which corresponds to the machine and detects the falling part, can be used.

1.5.2 Infrared Mold Protection

When using multiple-cavity molds, infrared protection beams are used. The transmitter and receiver are installed on the mold so that the measuring beam reaches the receiver just above the core. When the part remains on the core after a disturbance occurs, and the measurement beam is interrupted, the machine stops and triggers an alarm signal. Depending on the number of cavities, additional infrared beams per mold are required.

The mold protection protects the mold from damage during closing caused by molded parts that are not demolded and get stuck in the areas of mold cavities or parting planes.

1.5.3 Vision Systems

Vision systems are available that inspect in real time whether all the parts are ejected or whether sprues are stuck in the mold. These systems can not only take over ejector operations, but a variety of other monitoring tasks and control functions. Possible control options are:

- detection of surface defects, such as streaks, burns, and incompletely filled injection molded parts;
- precision dimensional accuracy checks; and
- assessing whether an extraction device accesses at the right place and whether the gripper securely holds the part.

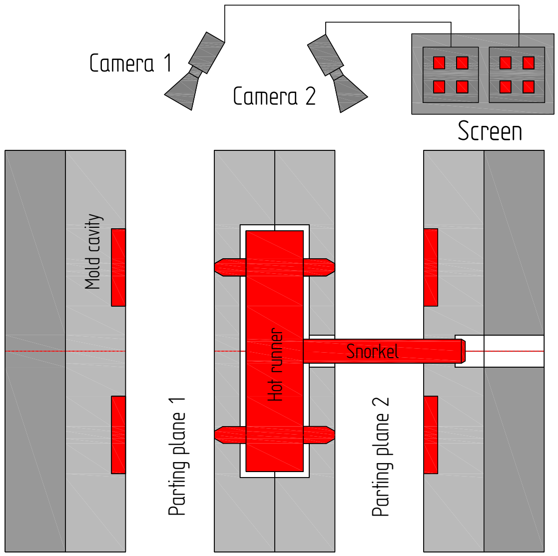

Figure 1.18 Principle of the vision system

Vision systems can take over a variety of monitoring functions without losing cycle time.

1.6 Mold Cavity Pressure/Mold Filling Control

During filling of the mold cavities, there are three phases with three different pressures: the dynamic phase, the quasi-static phase, and the static phase.

In the dynamic phase, the mold cavity is filled volumetrically. The mold filling pressure is adjusted at the end of the mold filling. This pressure is usually dependent on the injection molded part and the quality of the plastic material.

Next, the filling of the mold passes into the quasi-static phase, and the holding pressure takes effect. This causes a compression of the melt and compensates for the shrinkage. During this phase, the holding pressure is kept constant. This phase is in effect until the sealing point is reached. The sealing point indicates that the sprue at the gate is solidified and nothing is left to be pressed in. After the sealing point is reached, the pressure is reduced, and the shrinkage begins.

About 60 – 70 % of all errors in injection molding can be verified by weighing the injection molded part. Modern injection molding machines can monitor the injection process. There are two methods to detect the pressure in the mold. In the direct measurement, the pressure is recorded in the mold cavity. In the indirect measurement, the pressure is transferred to the pressure transducer through a mold element, usually an ejector pin. This pressure transducer is installed under the ejector pin.

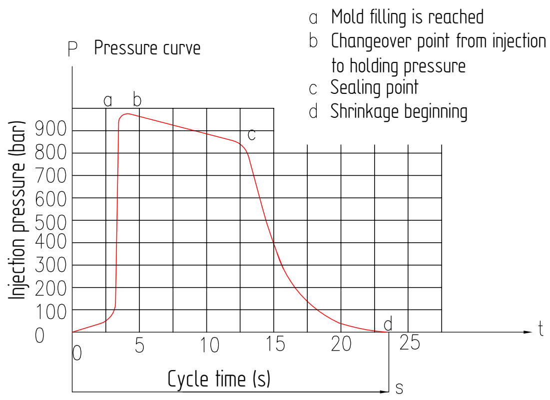

Figure 1.19 Pressure curve of mold cavity pressure

The mold cavity pressure is dependent on the factors such as the viscosity and solidification behavior of the melt, the geometry of the molded parts, arrangement of the cavities, and condition of the mold. There are two methods to detect the pressure in the mold: the direct method and an indirect measurement with a pressure transducer.

1.7 Simulation of the Filling Process (Moldflow Analysis)

The full observation of mold filling process can be done with a simulation program on a computer screen. The common Moldflow programs have a high accuracy in predicting the outcome of the molding process and help to avoid design errors.

A mold flow analysis calculates the material flow in the molded part. It shows where problem areas occur in the material flow. Critical points are the development of shear heat, the stalling or freezing of the material flow, or the formation of air pockets. The analysis also indicates the points at which heat must be removed, or where to optimize the material flow by controlling the temperature of the mold.

When the design is done using CAD systems, the mold flow analysis can be performed in parallel with the development of the design. The effects from changes in the wall thickness and the location of the injection point can readily be analyzed on the screen. By repeatedly changing the parameters, one can get closer to a molded part that will be produced with optimized technical flow properties.

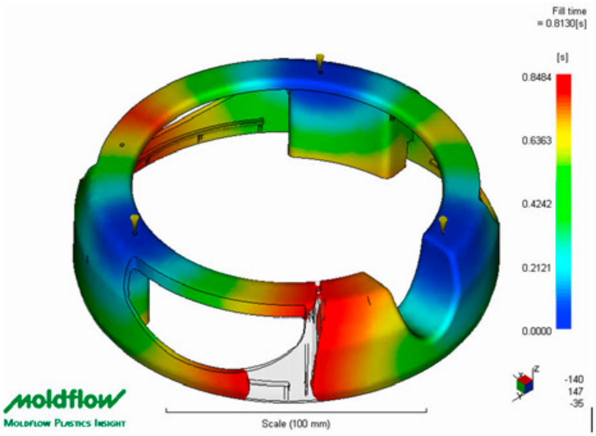

Figure 1.20 Moldflow-Analysis at three injection points (Source: FOBOHA)

Moldflow analysis allows a very early detection of problem areas during the development of an injection molded part and to produce an optimal mold.

1.8 Demolding Force

During demolding of a molded part, there are two types of forces: the opening force and the demolding force.

Demolding Force

The demolding force is the force required to demold the molded part. This force is composed of a break-away and an extension force. While breaking away, small undercuts caused by machining marks, structuring, and static friction must be overcome.

The reason for an extension force that high is often the fact that the draft angle is too small. It should be at 0.5 – 3°.

The rigidity of the cavity should also be considered. If it is not taken into account, expansion of the mold cavity during the injection phase can occur. In the cooling phase, when the holding pressure decreases, the core and cavity return to their original position. The consequence is that the molded part is clamped and can then only be "released" by disassembling the mold.

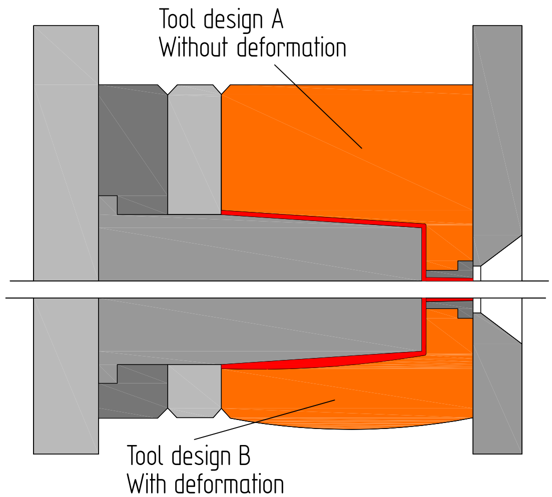

Figure 1.21 Principle of the demolding force

Opening Force

The opening force is created when the mold is opened, when the internal mold pressure can not degrade completely and therefore opens under residual pressure. The opening force of an injection molding machine is about 10 % of the clamping force, while the ejector force is at 3 – 5 % of the clamping force.

The strongest influences on the demolding force are the pressure profile during the injection, too-short or too-long cooling times, and an unacceptable mold deformation.

1.9 Ventilation

1.9.1 Ventilation by Displacement

The air in the mold cavity has to be displaced during the injection.
If the air cannot escape, it creates a back pressure, which complicates the filling process.

Especially for thin-walled, high-speed parts, an accumulation of air may occur when injecting that hinders the filling of the molded part. This makes a strong temperature rise possible.

Part of the ventilation is already created by segmenting the mold, installing cavities, sliders, and cores. But this is often not enough, particularly for thermoset parts. Therefore, the parting line should be lowered by 0.1 mm around the mold inserts, and a margin of 10 – 12 mm should remain around the inserts. The ventilation channels should be designed to be approximately 15 % larger in thermoset molds.

Another variant is to install ventilation channels with a depth of 0.2 mm and a width of 5 – 6 mm around the inserts. The depth of the ventilation channel to the mold cavity is 0.01 – 0.02 mm. These channels need a connection to the outer edge of the mold.

A decrease in the ventilation activity can be noticed by the fact that the mold-filling time extends or that the parts have flaws or burn marks in the areas of confluences. If this occurs, the ventilation is clogged and must be cleaned. The imprint of the mold cavity surface is better with good ventilation.

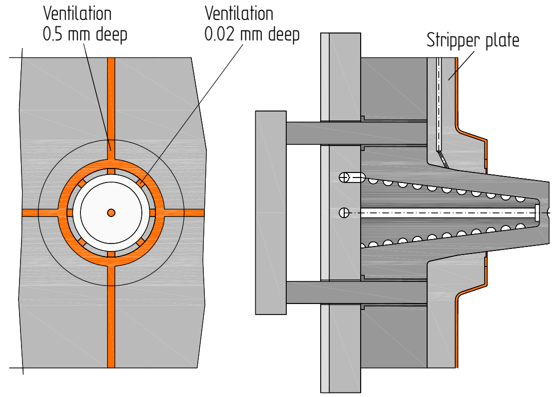

Figure 1.22 Principle of ventilation by displacement

To fill a molded part with a weight of approximately 500 g in one second means that about 500 ml of air has to be to be displaced. By this compression process, the air can become so hot that it burns the part.

1.9.2 Ventilation by Vacuum

When creating a vacuum in the mold, it must be sealed with an O-Ring. An evacuation is especially necessary in elastomer and thermoset processing for molded parts with partially very thin walls.

The venting of the molds with a vacuum actually began with the development of low viscosity plastics. Vaccum venting particularly pertains to molds that are used in liquid silicone and elastomer processing. Silicone is so low in viscosity that even ventilation channels of 0.01 mm depth would lead to over-molding. Therefore, the air needs to be sucked out of these molds.

To suck out the air from the parting plane, the mold must be sealed against the in-flowing air. The sealing is usually done with an O-Ring or a Quad-Ring. In addition, the ejector boxes and ejector bolts must be sealed. This ensures that a vacuum builds up.

In small molds, a constantly operating vacuum pump is used. The vacuum begins to build up once the sealing element is clamped in the two closing parting planes. In large molds, the pump switches on through a waypoint. Once the vacuum is built up, the machine starts the injection.

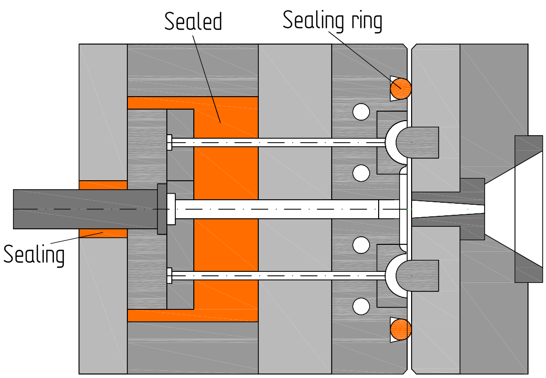

Figure 1.23 Principle of a mold with vacuum

The quality of the molded parts will determine whether ventilation by vacuum is required. Vacuum ventilation is recommended in molds for liquid silicone processing, and often mandatory for elastomer processing. Another advantage of vacuum ventilation is that a vacuum mold can be filled more quickly.

1.10 Support Bars, Support Plates, and Support Rollers

Support bars and support rollers have the function to intercept buoyancy forces and to support the mold against deflection.

The buoyancy force in the mold must generally be set equal to the closing force. This results in a deflection in the mold due to insufficient support of the mold plates. The deflection can cause overmolding on the molded part. For large mold inserts, breakage of the mold insert by the buoyancy force is also possible. If the buoyancy force acts centrally on the mold, there is a particularly high risk of deflection.

Support rollers or blocks take over the support in the central mold area. In support plates made from tempered mold steel, the support roller can be directly used against the pressure plate. If the mold steel is not tempered, inserting a hardened pressure plate into the area of support is recommended. In continuous operation, the use of support rollers is risky because multiple load changes can make the roller ineffective.

Injection molding machines that are used to produce buckets or thin-wall packaging often have reinforced machine clamping plates and smaller centering bore holes to counteract the deflections. The mold plate takes almost no support functions. All forces acting on the cores and dies are transferred to the support plate.

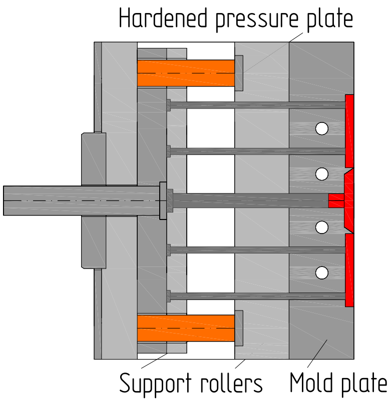

Figure 1.24 Principle of support rollers

Support rollers or plates take over the support of the mold in the central mold area and prevent deflection, which can result in over-molding of the molded part.

1.11 Mold Clamping Plate and Centering Ring

1.11.1 Mold Clamping Plate

The clamping plates are used for clamping and fixing the mold. In many cases, the clamping plates protrude from the narrow sides of the molds. This protrusion of 25 mm is used to turn off the exposed mold. To increase the free space, stabilizing rollers are sometimes screwed into the clamping plates instead of the protruding mold clamping plate under the mold.

If the molds are not equipped with thermal insulation plates, it is advisable to brush the plates with grease. An oil paper is pressed again this grease and then grease is again applied and brushed on. This prevents a formation of gratings between mold and machine clamping plates.

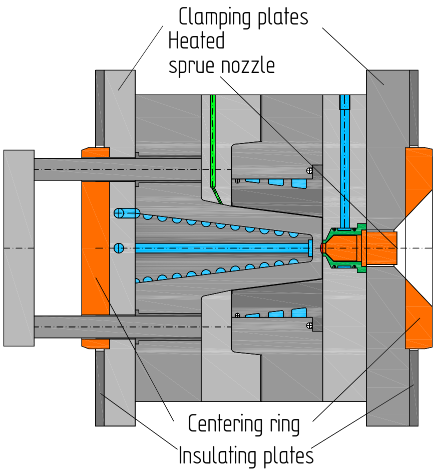

Figure 1.25 Mold clamping plate

1.11.2 Centering Ring

If a mold is not perfectly centered, the machine nozzle is working one-sided. The sprue bushing of the mold is not aligned with the machine nozzle, and melt can laterally escape during injection. It makes sense to only center on the nozzle side and to design the centering ring about 0.1 mm smaller on the ejector side. Thus, the centering ring only has a supporting function on the ejector side when installing the mold.

In order to facilitate the installation of the mold, it is advantageous that the centering ring has a distinctive lead-in chamfer.

The mold clamping plates are used to fix the mold on the machine clamping plates. The centering ring allows a precise centering of the mold with the machine nozzle.

1.12 Core Pullers

Core pullers (valves) are part of the equipment of the machine. They control components that belong to the mold.

Core pullers are needed to move the hydraulic cylinders, which in turn move sliders or jaws. They are also needed to operate hydraulic motors, which rotate rotary tables or drive a transmission, which rotates the threaded cores.

For movement in the mold, it makes a difference whether oil or air is used. Due to the fact that air is compressible, the movements in the air cylinders are often jerky. Therefore, oil is usually chosen as the driving medium.

In mold design, the function sequence is described and the core puller function is defined, which will later be programmed when used in the control of the injection molding machine. It is important that the final positions of sliders and jaws in the mold are monitored with end switches.

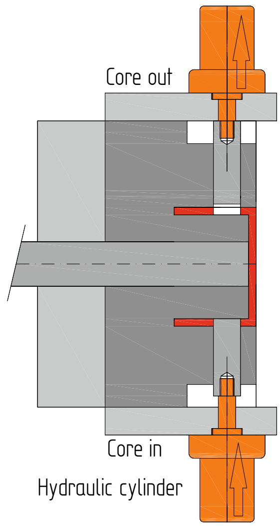

Figure 1.26 Principle of core pullers

Core pullers have the task of controlling hydraulic cylinders, which in turn move the sliders, jaws, and hydraulic motors or gears.

1.13 Mold Structure in Elastomer Processing

The injection of hot, vulcanized elastomer parts has become one of the most important processes in the elastomers sector. For producing elastomer molds, much can be taken from the long experience with thermoplastic molds.

Sprue and gate systems in elastomer molds are identical to those in thermoplastic molds. However, the cross sections of the manifold and the runners are somewhat smaller than for thermoplastic molds. The reason for this lies in the frictional heat that arises in narrow cross sections. This heat can be used so that the mass is heated faster and thus flows better and cross-links faster.

Completely different is the situation with the mold temperatures. They are, depending on the elastomer, between 170°C and 200°C. The molds are generally electrically heated. If molds are often changed, hot plates, which remain on the injection molding machine, are used. Therefore, a heating in each mold can be avoided.

Elastomer molded parts can generally be very easily deformed. Therefore, the ejector pins are not sufficient for demolding. Demolding by hand, with handling equipment, brush-off units, or by using compressed air, is therefore common.

Reference

1. Stitz, S., Keller, W., *Spritzgießtechnik, Verarbeitung, Maschine, Peripherie*, (2004) Hanser Publishers, Munich

Hot runner technology is not used in elastomer molds. If multi-cavity molds are used, it has to be ensured that the flow paths are all the same length (balanced) to the central nozzle. This is important so that the filling of the mold cavities is the same.

2 Types of Demolding in Two-Platen Molds

2.1 Molded Parts without Undercuts

Molded parts without undercuts are parts that show no resistance and can be ejected without effort when opening the mold and ejecting the molded parts.

In the field of plastic processing, there is a variety of applications from the "parts without undercuts" sector. Demolding without undercuts is used in nearly all mold techniques, except in three-platen molds. Molds for parts without undercuts are relatively simple "open-close" molds without additional movable components.

To be able to easily and safely demold the sprue, it has to be shaped conically and should be designed with a conicity of about 3°.

Typical applications include injection molded parts that can be released and ejected without an additional parting plane or additional movements. These parts include, for example, mugs, caps, covers, door handles, and clothes hangers.

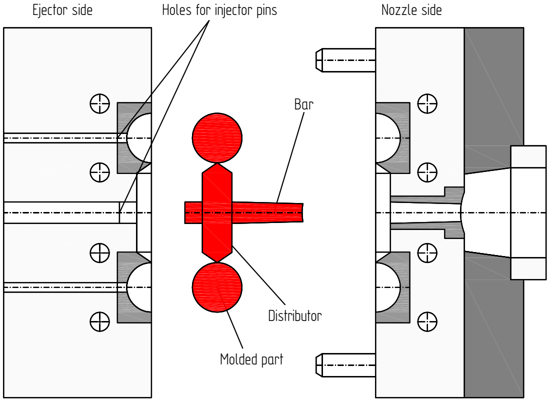

Figure 2.2 Principle of demolding without undercut

Figure 2.1 Demolded parts without undercut (Source: Ferromatik Milacron)

The characteristic of this type of mold is that the molded parts and the sprues are ejected from the common parting plane of the mold.

2.2 Molded Parts with Undercuts

2.2.1 Elastic Stripping

A positive or negative change in the geometry at the inner or outer diameter of an injection molded part is called an undercut. Parts with internal undercuts are fundamentally harder to demold.

The term elastic stripping only applies when the molded part is demolded by a simple stripping during the opening movement of the mold without requiring a slider. To make this possible, the material must be tough and elastic. Ideal materials include polypropylene (PP), polyethylene (PE), soft polyvinyl chloride (PVC), or elastomers.

The demolding is usually done through a so-called stripper plate. Ejector pins, push-out sleeves, or air may also be used. Molded parts are usually demolded without additional ejector elements.

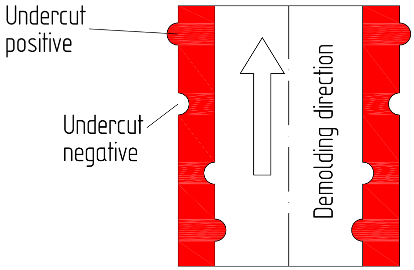

Figure 2.3 Principle of an undercut

Application

Most applications can be found in the area manufacture of screw caps. Many applications are also used in the medical and packaging area.

Figure 2.4 Back-injected part, demolded (Source: FOBOHA)

Undercuts are depressions or recesses on the injection molded part that complicate the demolding. Under certain conditions, depending on the material and geometry, a demolding by stripping the elastic part is possible.

2.2.2 Demolding through Sliders

Undercuts are depressions or recesses on injection molded parts that complicate the demolding or even make it impossible in extreme cases. Auxiliary parting planes can be generated with sliders to release partial molded part areas that have an undercut in the demolding direction.

Slider Mold with Cold Runner

Sliders are incorporated either into the nozzle or ejector side and move transversely to the opening direction of the mold. When the mold is opened, the slider runs to the side of the mold axis at an angle of 90°. A maximum deviation of 7° in both directions should not be exceeded or the slider tends to lock.

The movement of the slider is carried out mechanically by a sloping or control bolt, also called the positive control.

The movement of the slider can take place before, during, or after the opening of the mold, either mechanically or by means of springs, air, or hydraulic cylinders. When determining the dimension of sliders, the buoyancy force that arises in the mold has to be considered (see Section 1.3).

Typical applications include power strips, bobbins, clamps, dowels, and parts with external threads.

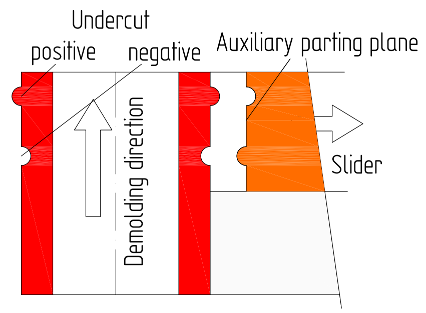

Figure 2.5 Principle of an undercut with slider

Figure 2.6 Injection molded part demolded with slider (Source: Ferromatik Milacron)

In the slider molds, a slider runs out transversely to the mold axis when opening the mold. Thus, the molded part is released and the stripping is enabled.

2.2.3 Demolding with Jaws (Split Molds)

In split molds, two or more jaws completely enclose the molded part, in contrast to slider molds where only parts are released.

In slider molds, the mold cavity usually consists of one part. However in split molds, the jaws form the mold cavity. The jaws can be inserted diagonally on the nozzle side and they then move on the diagonal to the outside when the mold is opened by means of a pull tab. Thus, the injection molded part is released for demolding.

Alternatively, the jaws can also be guided on the ejector side, as with sliders. They are then moved, during or after the opening movement, mostly with hydraulic cylinders or mechanically using springs or air.

Jaws in all sizes should be necessarily integrated into the cooling circuits of the molds.

Split molds can be combined with all known manifold systems and gate variations. The sprue and the molded part fall out of the same parting plane and need to be sorted later.

A very typical application for split molds is the manufacture of bottle crates and car batteries.

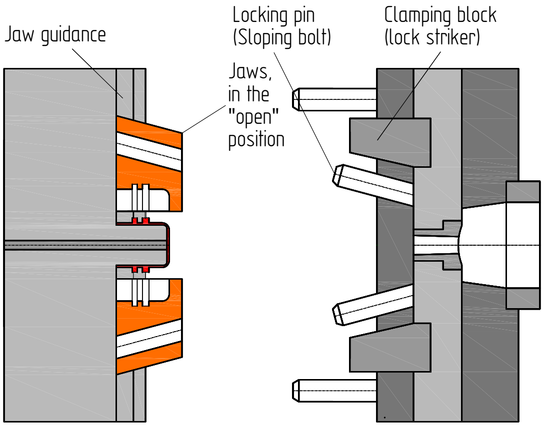

Figure 2.7 Principle of a split mold

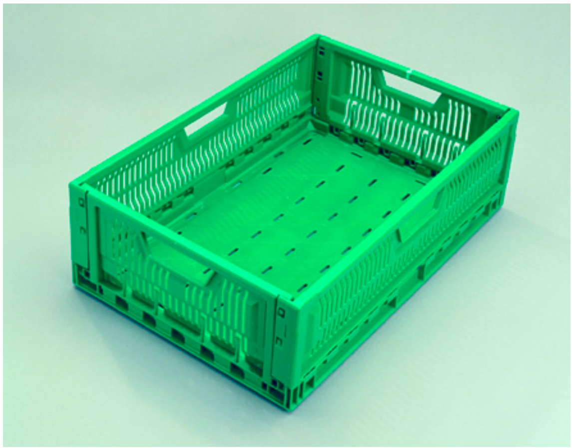

Figure 2.8 Injection molded part demolded with jaws (Source: Ferromatik Milacron)

When starting up the machine, the jaws of the mold can be opened via a slope and the molded part is released for demolding.

2.2.4 Collapsible Cores

Collapsible cores are used when, besides threads, other undercuts must be released. These parts include tamper evidence bands, tear tabs on bottle caps or the like.

When using collapsible cores, one must ensure that the thread depth and the thread diameter are correlated. The thread depth is about 0.8 – 1.0 mm at a thread diameter of 28 mm. Therefore, no thread with a diameter of 28 mm and a thread depth of 3 mm can be produced with the collapsible core. However, this would always be possible using a rotary core.

This interdependence of thread depth and thread diameter results from the fact that the segments, which are arranged around a central core, can only fall so far inward until they lock themselves when the central core is pulled.

Applications can be found wherever the thread or a thread-like undercut on the molded part plays a minor role, such as covers used for resealable containers.

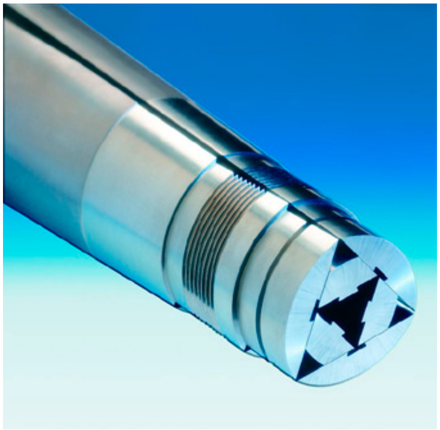

Figure 2.9 Collapsible core (Source: HASCO)

In general, threads or thread-like undercuts on injection molded parts with a diameter of between 15 mm and 500 mm can be produced with collapsible cores. When using collapsible cores, the plastic material being used is irrelevant.

2.3 Molded Parts with Internal and External Threads

Considering the demolding direction, internal and external threads represent undercuts. These parts are demolded by rotating the threaded cores (for internal threads) or by turning the dies or cavities (for external threads).

2.3.1 Internal Thread

Internal threads are located on a core with a thread profile, which is encapsulated during the injection. The molded part is smooth on the outer surface.

If parts with internal threads cannot be demolded elastically, demolding must be done by unscrewing or with collapsible cores.

Figure 2.10 Injection molded parts with internal threads (Source: Ferromatik Milacron)

2.3.2 External Thread

External threads are usually created with jaws. The parting plane may not be visible, and the entire cavity is rotated.

For rotating the threaded cores or the cavities, a gear is installed in either the nozzle or ejector side. The gear is either driven by a servomotor, a hydraulic motor, a rack, or by a helical spindle that is pulled by a lead screw.

For both external and internal threads, the molded parts are held through the fluting on the periphery of the part or through specially attached twisting jaws. In the latter, the twisting jaws are often visible on the inside of the part. If this is not desired for the finished part, unscrewing must be done when the mold is closed.

2.3.3 Unscrewing Device

To be able to unscrew the threaded core, the molded part must be secured against rotation. Several variants are available, depending on the shape of the molded part.

2.3.3.1 Types of Unscrewing

2.3.3.1.1 Unscrewing when the Mold is Closed

The screw cores are controlled by a guide thread and retracted. It should be noted that the guide thread has the same incline as the thread that is screwed out. Protection against rotation comes from the molded part itself, which is held in the nozzle side by its outer geometry. After unscrewing, the machine drives up, and the parts are ejected by the stripper plate.

Advantage: Molded parts are molded without damaging the last thread.
Disadvantage: The threaded core has to be rotated forward in spraying position, which will increase the cycle time.

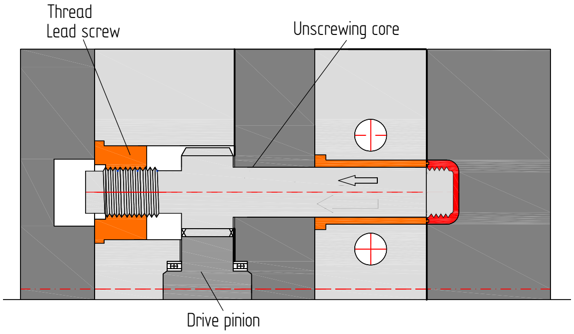

Figure 2.11 Principle of unscrewing when the mold is closed

2.3.3.1.2 Unscrewing for an Attached Stripper Plate

The stripper plate is attached to the nozzle side. The rotating cores are not retracted. The machine opens parallel to the thread pitch. The stripper plate is pressed against the nozzle side with spring force. The injection molding machine drives up and the parts are ejected by the stripper plate.

Advantage: No expensive design of the threaded cores or expensive lead screws are required. No loss of time occurs by turning back the threaded cores.
Disadvantage: A portion of the closing force is lost by the counter force of the springs.

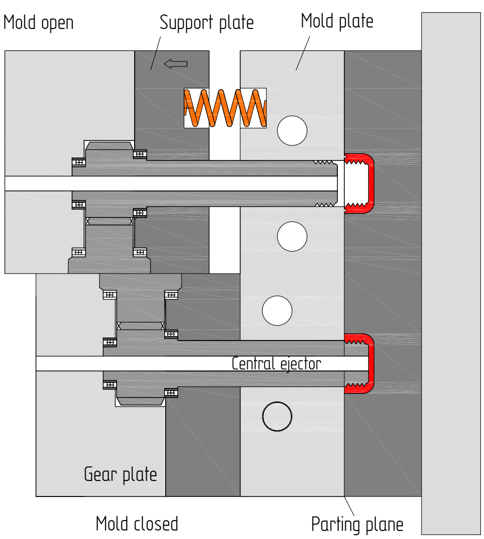

Figure 2.12 Principle of unscrewing for an attached stripper plate

2.3.3.1.3 Unscrewing during Upward Motion

The molded part is attached to the stripper plate. Special retaining cams prevent the molded part from rotating. In this variant, the screw core is brought back. It eliminates the press-on springs. The core, guided by the guide thread, rotates back and releases the parts.

Advantage: The injected threads are of very high quality.
Disadvantage: The manufacture of the threaded core and the lead screw is expensive.

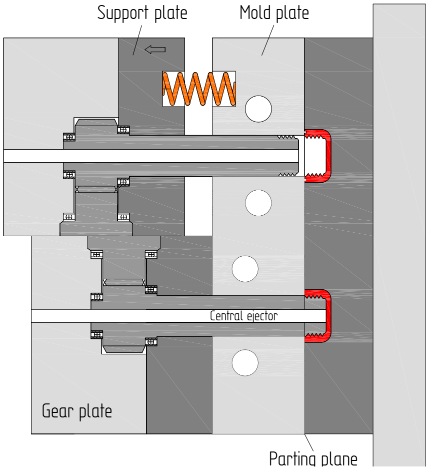

Figure 2.13 Principle of unscrewing the stripper plate with spring force

2.3.3.1.4 Unscrewing the Stripper Plate with Spring Force

The cores do not run back; they are pressed back with the spring force of the support plate. The molded thread in the injection molded part takes over the task of the lead screw. Here, the recently molded thread is strongly loaded. There is the danger that the last gear can be damaged upon leaving the core from the threaded part.

Advantage: The manufacture is cost-efficient.
Disadvantage: The thread is of lower quality.

In unscrewing molds, the molded part is demolded by rotating the threaded cores (internal threads), or the cavities, also called dies or mold cavities (external core).

2.4 Molded Parts with Threads, Forcibly Demolded

In forced demolding of threaded parts, the molded parts are stripped by a stripper plate. Prerequisite for this is on the one hand, the use of tough, elastic, plastic masses and on the other hand, molded parts that have rounded thread crests.

Typical applications for forced demolding are screw caps for the beverage industry. Around 80 % of the world's billions of caps required annually are forcibly demolded. The rest are either unscrewed or the thread is released with the collapsible core.

Releasing with the collapsible core has the advantage that the molded tamper evident band or tear tab can be more easily demolded.

In injection molds with undercuts that are forcibly demolded by stripping, it must be ensured that the molded part is released. Therefore, the molded part can avoid the overcoming of the undercut and is not deformed. If the injection molded part is not attached to the outer diameter of the mold, demolding is possible.

Many years of experience in stripping technology are necessary to build molds that run 24 hours a day without problems for at least 300 days a year.

Due to the high parts output that is expected from these molds, these molds are used among the fast-cycling injection molding machines.

To ensure a safe demolding, the thread depth should not be greater than about 8 – 10 % of the part diameter. The plastic material should be elastic, but tough. PE and PP are especially suitable.

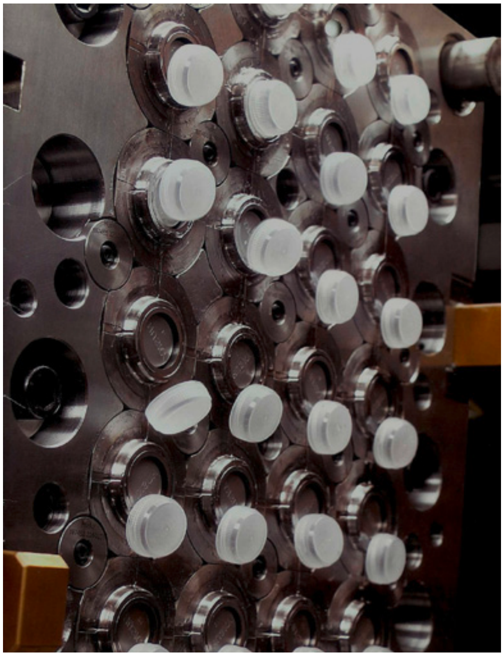

Figure 2.14 Parts with threads, forcibly demolded (Source: FOBOHA)

In forced demolding, elastic but tough molded parts are stripped with an ejector plate, which is coupled with the ejector of the injection molding machines.
The connection to the machine has the advantage that a vibration function can be executed with the stripper plate. This ensures that all parts are molded.

2.5 Unscrewing Gears

An unscrewing gear is required whenever gears or gear-like devices must be installed into the molds. Examples are unscrewing molds or two-component molds that have a mold-internal rotary drive.

In an unscrewing mold, the gear, which is integrated in the injection mold, must be primarily designed. To determine the gear, the following information is required:

- Thread length (number of rotations)
- Rise and number of core rotations
- Diameter of the threaded part
- Material (to determine unscrewing force needed)
- Number of cavities (reference circle determinative)
- Cycle time (rotational speed)
- Parts are arranged in series or around a central pinion.

Using this information, the gear is designed and calculated. In this respect, it must only now be decided how the gear should be driven.

If the parts are in series and the thread is short, a hydraulic cylinder, which moves a toothed rack back and forth in the mold, can be used for unscrewing. The length of the rack limits the number of rotations.

If the screw cores are located around a center gear and the thread is short, a helical spindle that rotates the center gear when opening the machine can be used for unscrewing.

The gears can be driven by a servomotor or a hydraulic motor. The servomotor drive is becoming increasingly important. It is very precise to control and oil-free.

3 Gate Technology

3.1 Distribution Systems

In the mold design, determining the gate location and choosing the type of gate is of the utmost importance for the economics of injection molding.

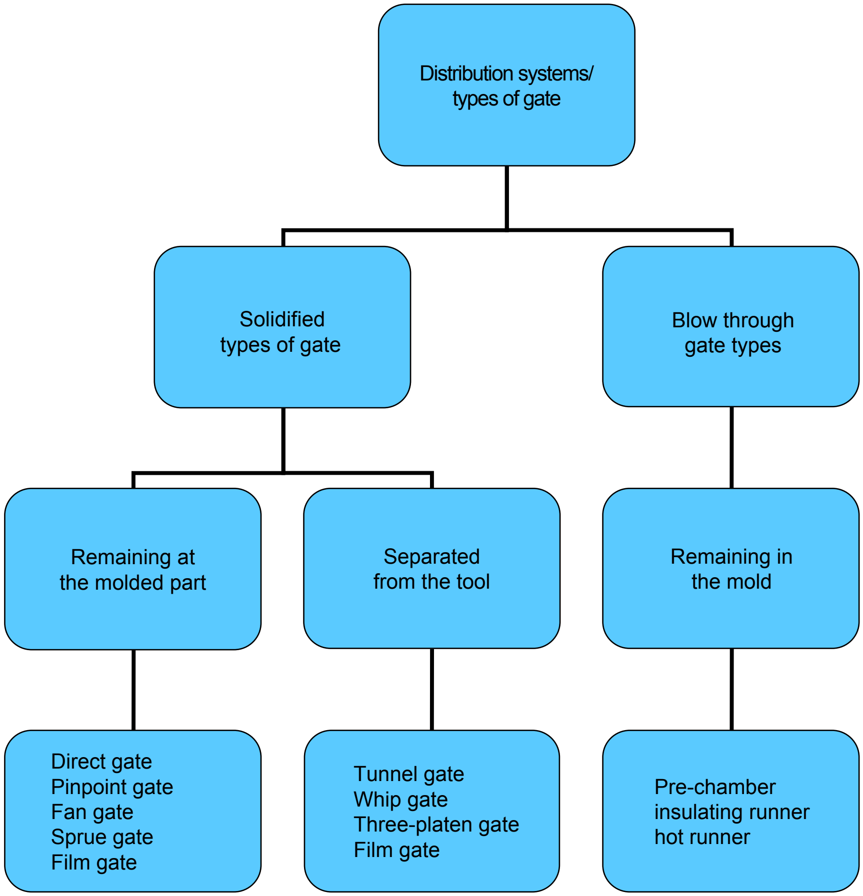

Table 3.1 Overview of the Types of Gates/Distribution Systems

3.1.1 Distribution System with Demolded Molded Part

3.1.1.1 Cold Runner

In cold runner molds, the injection molded part and the gate are cooled in the mold. They are ejected together in each cycle.

The design of the cold runner is implemented so that all cavities are filled uniformly. Here, an equal distance of all cavities to the central injection point is important to ensure that the mold filling proceeds uniformly.

If the cross sections of the gate rods or distributor arms are too thin, high injection pressures must be used to fill the injection molded parts. High filling pressures lead to stresses and stress cracks in the molded parts.

If the pressures are too high, the internal mold pressure increases. The injection molding machine has no more clamping force reserve, and the injection molded part is over-molded as a result.

To determine the cross sections of the distribution channels as a function of the weight of the molded part, reference values are determined from practice.

The diameter of the runner should always be at least as large as the largest wall thickness range in the molded part.

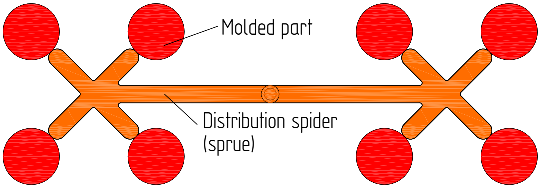

Figure 3.1 Distribution spider, 8 joints

Table 3.2 Distribution Channels Versus the Weight of the Molded Part

Parts	Weight in g	Duct cross section Ø
Small	> 10	3 – 4 mm
Medium	11 – 30	4 – 5 mm
Large	> 30	5 – 6 mm

The design of cold runner molds can easily be achieved. Distributors and gates are easy to produce. Subsequent separation of molded part and gate is necessary.

3.1.1.2 Three-Platen Distributor

Three-platen molds have the advantage in that molded parts and sprue spiders are ejected from two different opening areas (parting planes) of the mold. This eliminates an additional step, namely, the separation of the gate from the molded parts.

The three-platen system provides the ability to fill the parts through one or more sprues. This means that the design freedom is more diverse than in two-platen molds. The three-platen distribution system operates with a tear-off pinpoint gate.

When the mold drives up, retaining pins, which were injection molded, or retaining undercuts hold the gate on the fixed mold half. As a result, the gate tears off from the part. An additional mold function on the nozzle side, similar to the ejector, ejects the distribution spider. An advantage of three-platen molds is the very close cavity spacing.

It should be noted that the distributor cross sections are adapted to the molded part weight and that the distribution spider is balanced. Through the ability to open the mold in two planes, opening of one of the planes can be delayed until, for example, the mold cores are pulled out of the molded part or the thread cores are unscrewed.

Opening of the three mold packages, which consist of nozzle, central, and ejector plate, is mostly performed by the driving up movement of the machine. Latches or pull-tabs determine the path to drive up. Air or hydraulic cylinders are also often used to open the packages. The advantage is that the point of opening the mold packets can be controlled by time.

The distribution spiders are often the cause of production problems. To be able to remove them trouble-free from the failure area under the mold, a gate removal robot can often be used. These robots are often programmed to directly supply the gates to a plastic mill.

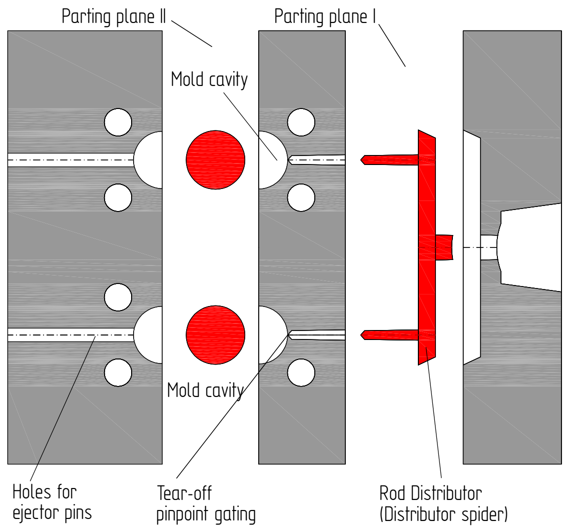

Figure 3.2 Principle of a three-platen mold

Three-platen molds are always used with a cold runner system. The molded parts are injected with a tear-off pinpoint gate. A material or color change is easy to perform. The sprue, which is often a problem during ejection, may be replaced by a heated nozzle.

3.1.2 Distribution System, Remaining in the Mold

3.1.2.1 Insulating Runner

Molds with distribution systems that remain in the mold, are either thick in cross section to ensure that the plastic remains fluid in the core of the distributor (the insulating channel), or the distribution systems must be electrically heated through a hot runner system.

Single-cavity molds are directly injected to the part with a rod, a pre-chamber, or a heated nozzle. Multi-cavity molds need a distribution system.

Originally, only cold runner distributors were used. Later, molds with extremely voluminous distribution systems, which were molded into a second mold plane, were added. These gates, called "bones," remain in the mold during production. This only works because the thick runners insulate themselves, hence the name insulating runner mold.

In case of disturbances or material/color changes, the second mold level is opened and the runner is removed.

Insulating runners are often used for high-speed injection molding parts. Depending on the system, a cycle time of 15 sec. must not be exceeded.

Typical applications are yogurt containers, screw caps, disposable cutlery, disposable syringes, pens, and plant pots.

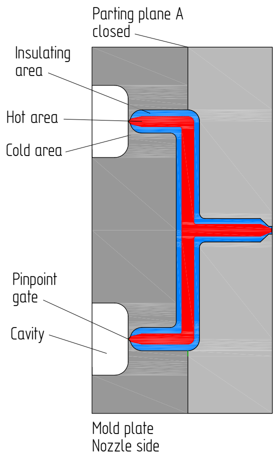

Figure 3.3 Closed insulating runners

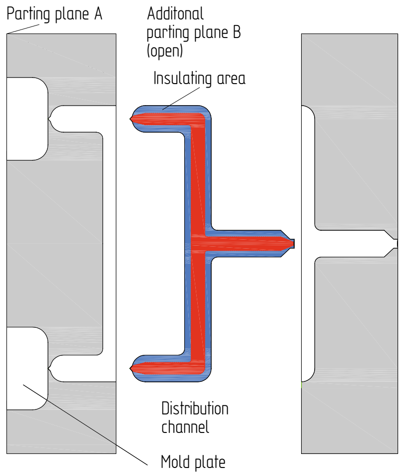

Figure 3.4 Open insulating runners

Compared to hot runner molds, insulating runner molds need a higher clamping force, but no electrical energy.

3.1.2.2 Hot Runner

3.1.2.2.1 Advantages of a Hot Runner

The hot runner is theoretically an extension of the injection unit. The hot runner should keep the liquid molten up into the mold cavity. It meets this requirement by guiding and heating the molten plastic until it reaches the mold cavity, with virtually no temperature and pressure losses.

The biggest advantage of the hot runner technology lies in waste-free injection and in good injection point quality. Disturbances in the production process by stuck sprue spiders in the open mold do not exist.

Another advantage is the positioning of the injection point. It can be selected easily so that a balanced filling of the molded part is guaranteed. Symmetrical filling means that injecting is done in the middle of the base. This takes place, for example, in all cylindrical shapes such as cups, buckets or similar parts.

In irregular injection depths, all part surfaces are graphically projected onto one plane. A circle, which covers all subareas, is then drawn. If an injection in the center of this circle is possible, a symmetric filling of the molded part is ensured.

By eliminating disruptive distribution channels in the parting plane, design possibilities that are completely new for mold making can be obtained. The molds become more compact and the number of cavities in the molds is significantly increased. Molds with up to 128 mold cavities or more are not unusual.

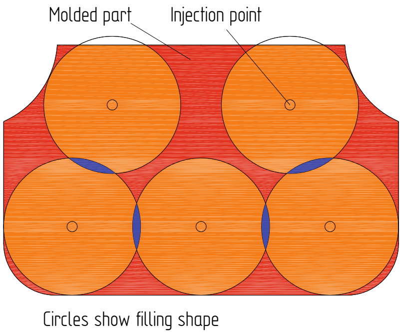

Figure 3.5 Principle of symmetrical filling

The hot runner technology guides the melt to the cavity with nearly no loss of temperature and pressure. Injecting without a gate eliminates the use of a sprue spider, which causes a substantial material and cycle time reduction.

3.1.2.2.2 Hot Runner, Internally Heated

Molds with internally heated hot runners belong to the beginnings of the hot runner technology and are now hardly ever built. A disadvantage of internally heated systems is the complexity involved when changing color or material.

The hot runner technology originated from the idea of developing an internally heated distribution and gating system from an insulating runner, which prevents solidifying of the runners when starting the mold. Therefore, a heating system was installed and placed in the center of the distribution channels.

The knowledge that no hot material can escape in places where cold, solidified material is located was an important condition for the tightness of the system. A big disadvantage occurred for color or material changes. Streaks formed on the molded part because material particles dissolved in the surface layers of the insulating layer. To rule this out, the entire system had to be disassembled for every color and material change and the material located in the distribution system had to be removed, which is a very complex procedure.

The heating system is located in the center of the mass flow. The heating elements are heating cartridges that regulate the temperature by means of sensors. The cartridges are almost self-insulating. The heat transfer into the cold mold is very low; this guarantees a low thermal expansion, and thus a high probability that the mold is sealed.

The internally heated system was preferably used for the processing of semicrystalline thermoplastics such as polyethylene (PE), polypropylene (PP), or polyamide (PA).

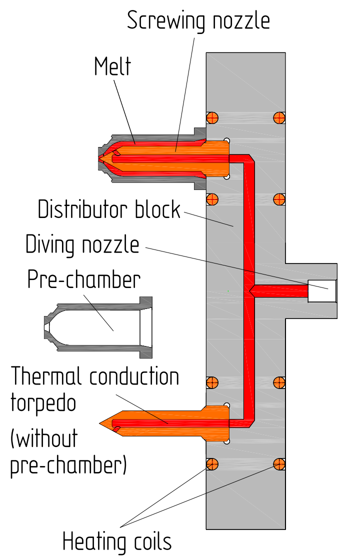

Figure 3.6 Principle of a hot runner system

The change of color and material is a major problem for internally heated hot runners. To still be able to inject perfect parts after a change, the distribution system is disassembled and material residue is removed. Therefore, the internally heated hot runner is hardly used today.

3.1.2.2.3 Hot Runner, Externally Heated

In externally heated hot runner systems, the heat is transferred from the outside into the distributor plate or beam. Here, the plastic melt in the distribution bore holes are not in contact with the heating elements.

Heating Coils

The heating elements are mostly form-bent heating coils. They are inserted on both sides in the distribution beams or plates in milled channels and are usually poured out correctly with two-component, heat-conducting cement. Sometimes, the heating coils are only covered with sheets. A uniform distribution of heat in the distribution system can be achieved with heating coils.

Heating Cartridges

Alternatively, heating cartridges, which are usually drilled into the distribution block close to the plastic receiver hole, can be used. These bore holes should be manufactured with an optimum fit to ensure a good heat transfer.

The heating cartridges are usually incorporated with thermally conductive paste. This causes a better heat transfer, and the cartridges are more easily exchanged, if necessary. Today, heating cartridges are only used for simple molds.

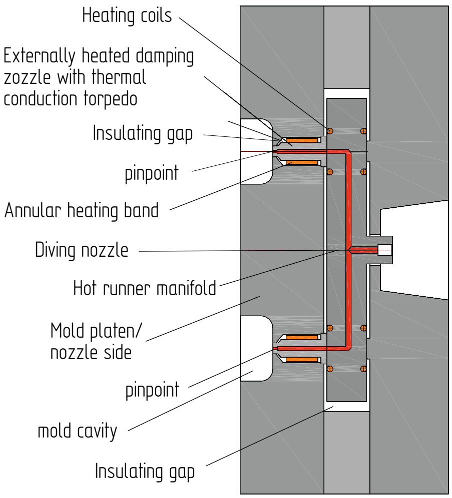

Figure 3.7 Principle of hot runner, externally heated

In an externally heated hot runner, the heat is introduced via heating of the distributor housing. The temperature above the flow cross section of the melt channel is approximately constant.

Temperature Control

Each heating zone is individually controlled by temperature sensors. If multiple temperature sensors are used in different heating zones, a contour-dependent temperature control is possible. The sensors are combinable with all common types of nozzles, such as clamp, screw-in, thermal conductivity, and valve gate or torpedo nozzles.

Whereas today the temperature of all the nozzles and the distributor block is usually controlled at only one location, in older molds, only the distribution block was controlled in the middle by means of a temperature sensor.

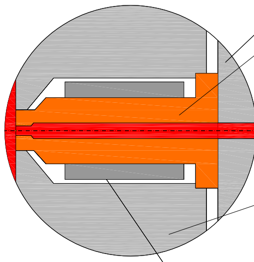

Figure 3.8 Hot runner clamping nozzle

Thermal Expansion

Of particular importance is the controlled thermal expansion in the hot runner distributor. There are several ways to compensate for this expansion. In the first method, the distribution beam is not directly connected with the nozzle; during the expansion, the distribution beam slides across the nozzle. In the second method, the thermally conductive nozzles are screwed into the distributor block. It should be noted that the distance between the nozzle bores holes (inside micrometer) is reduced by the amount of expansion.

As soon as the hot runner reaches operating temperature, the nozzles are, due to thermal expansion, in the center of the pre-chamber.

All hot runner systems have to be calculated in terms of their thermal expansion.

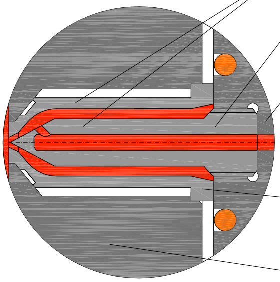

Figure 3.9 Hot runner screwing nozzle

The thermal expansion can be calculated using the following formula:

$WA = VL \times WF \times \Delta T$

WA = Thermal expansion in mm

VL = Distributor block length in mm

WF = Coefficient of thermal expansion

ΔT = Temperature difference in °C

3.1.2.2.4 Multiple Connections

There are three different types of multiple connections: 1) one nozzle for multiple injection molded parts, 2) multiple nozzles for one molded part, and 3) multiple nozzles for multiple molded parts.

One Nozzle for Multiple Parts

This type of nozzle, also called a hot edge nozzle, is designed such that multiple parts can be injection molded with one nozzle (see Section 3.2.2.3).

Multiple Nozzles for One Injection Molded Part

This variant, also called cascade injection molding, is the classic version of multiple connections. Large and extremely long parts, such as automobile bumpers, are molded with multiple nozzles. Even parts with different levels, such as garden chairs, are molded with different lengths of hot runner nozzles. This process is preferably used when no clamping force reserve is present on the part of the injection molding machine (see Section 6.9 Cascade Injection).

Multiple Nozzles for Multiple Injection Molded Parts

When using multiple nozzles for multiple molded parts, it is important that the flow paths from the hot runner to the distributor are approximately the same length in all nozzles to prevent a nonuniform filling.

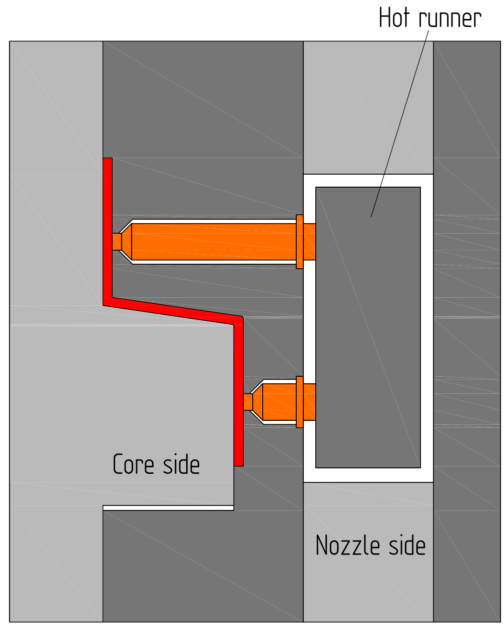

Figure 3.10 Principle of multiple connections

Multiple connections are especially used for very large molded parts. They allow molding, which is adapted to the injection molded part, with nozzles of varying lengths. Even with mass-produced parts, several molded parts can be connected with one nozzle for cost reasons.

3.1.2.2.5 Needle Shut-Off Nozzles

In general, needle shut-off nozzles operate through a hydraulic, pneumatic, or electric actuation. If a careful processing of the plastic melt is demanded, controlled needle shut-off nozzles are used.

Standard hot runner nozzles, whether with or without heat-conductive tips, always have a problem: The nozzle opening is either too small or too large.

If the opening is too small, injecting must be done at high pressure. The consequence is that high stresses occur in the part and the plastic is thermally damaged by the heat development at the nozzle tip. If the opening is too large, the so-called tear-off – the separation between gate and injection molded part – happens poorly; plastic material is drawn from the distributor and remains on the part as a gate flash.

The solution for both problems is a nozzle that can be opened and closed with a valve, called a needle valve.

A general distinction can be made between conical and cylindrical guidance of the needle.

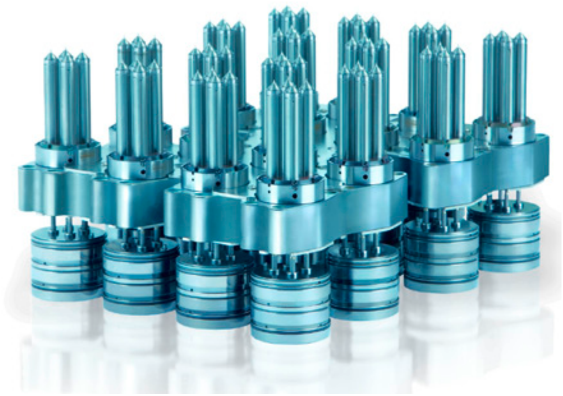

Figure 3.11 Needle shut-off nozzles (Source: Otto Männer)

Conical needle shut-off valves

The conical needle shut-off valves are usually very intricate in detail in the area of transition to the cavity. The remaining wall thicknesses are around 1 – 2 mm. As a result, the bottom of the nozzle can break through the closing movement of the needle. Problems can also occur during closing in the cone area, when material residues remain stuck in the needle. The result is that the nozzle does not close, and thus surface streaks occur on the molded part.

Cylindrical needle shut-off nozzle

With the cylindrical needle shut-off nozzle, it is possible to displace the residual material from the needle into the mold cavity without any problems. The needle is first centered during the closing movement through a conical guide and is exactly installed to the axis of the gate bore hole through a cylindrical guide. This guarantees a very low wear and a long service life of the needle because the closing movement of the gate is carried on contact-free.

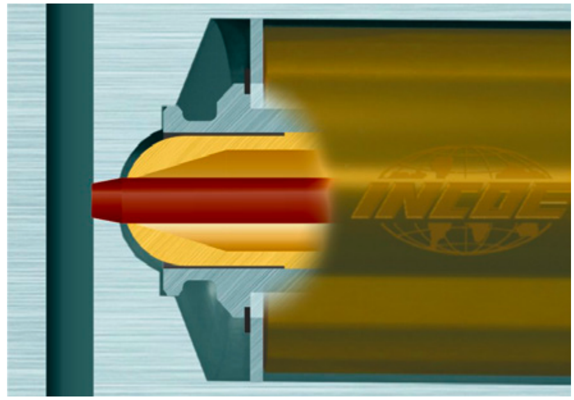

Figure 3.12 Principle of a conical needle shut-off nozzle (Source: INCOE® Corp.)

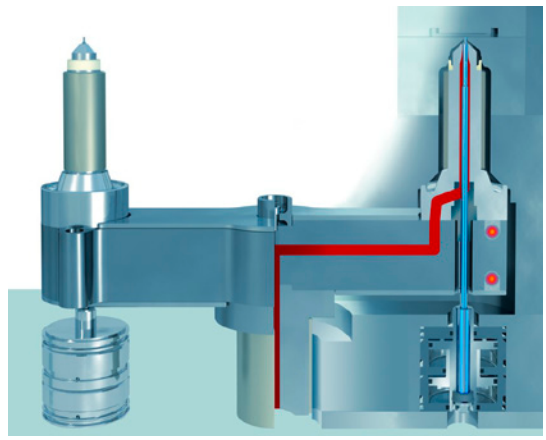

Figure 3.13 Principle of a cylindrical needle shut-off nozzle (Source: Otto Männer)

> The clean separation of the melt and the reliable closure of the nozzle during the dosing phase is the task of a needle shut-off nozzle. Today there are different locking mechanisms of the needle on the market.

3.1.2.2.6 Hot Mold Halves

The term “hot mold halves” is used only when multiple nozzle-sided mold halves with a hot runner are used for one mold.

In the hot runner, plastic melt is not cyclically exchanged, but parts of it remain in the hot runner system. This is quite different for the cold channel. Here, the runner is ejected in each cycle, which means that new material is always available from cycle to cycle.

If the material or the color is replaced in the hot runner, great effort is needed. Either the hot runner is heated and then rinsed with the new material or the system is completely disassembled, cleaned, and then started up again.

For each material/color, a “hot mold half” is used to reduce this time-consuming process. That means that the nozzle-sided distribution system is present multiple times and is accordingly replacement during production.

The “hot mold half” includes a completely wired and ready-for-connection system with clamping, frame and nozzle holder plates, and all the hot runner components. It just needs to be connected to the other mold components. This concept of “hot mold halves” shortens the design time of the mold maker considerably. As a rule, a complete system consists of a heating system, temperature sensor, temperature control circuits, and ready-to-connect electrical wiring.

Figure 3.14 Hot mold halves (Source: HASCO)

For a material change, the “hot mold half” is exchanged and new production can begin directly. Time-consuming disassembly and cleaning of the hot runner system is eliminated.

3.2 Gate Technology

The distribution system and the sprue have the task of guiding the liquid plastic melt into the mold cavity on the shortest route without heat and pressure losses. The right choice of the distribution system and the appropriate gate variant is one of the most important determinants of whether a project that consists of an injection molded part and mold can be successfully implemented.

The molded part determines the optimal combination of distribution system and gating technology. Knowing the production volume, reproducibility, accuracy, and material is essential for the proper selection. For small quantities, molds with a single cavity are generally used for cost reasons. Molded parts that require a high degree of reproducibility are also produced in molds with a single cavity. Thus, mold filling pressures (injection and holding pressure) can act for precision parts such as gears or similar technical parts, the optimal variant of the direct gate is necessary.

The sprue system has the task to accept the plastic melt of the injection molding machine without loss of pressure and temperature and to transfer it to the mold cavities.

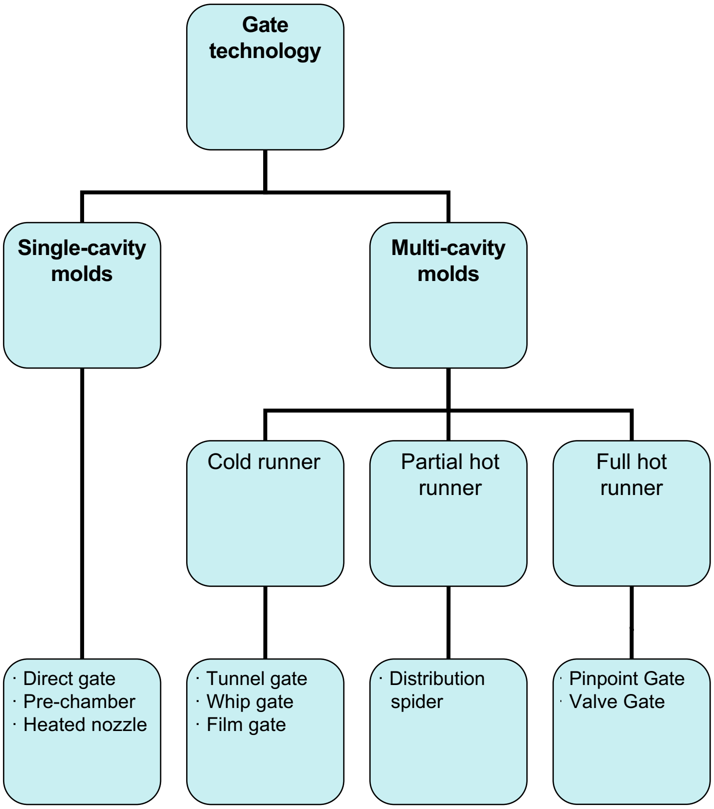

Table 3.3
Overview of Gate Technologies

3.2.1 Solidifying Gate, Remaining on the Molded Part

3.2.1.1 Direct Gate

Compared to all gate systems, the direct gate is the easiest to produce. However, the subsequent removal of the direct gate has become so expensive that it is today used only in special cases. The direct gate is not utilized now as often as in the past.

The direct gate is directly and centrally placed on the molded part. This ensures that the plastic melt flows directly into the mold cavity virtually without temperature and pressure losses.

In the direct gate, the bore hole for the melt feed in the bushing is conical; it widens towards the mold. In the design, as with all types of gates, it should be ensured that the wall thickness of the molded part is biggest at the gate and smallest at the flow path.

In practice, the thickest part of the direct gate is calculated as follows: average wall thickness of the molded part + 2.0 mm safety factor.

The direct gate must be subsequently removed mechanically, either by cutting or milling. The alternative is a heated gate nozzle with pinpoint gate. The rework is then eliminated, but a small pressure loss will occur. This must be considered when optimizing the injection process.

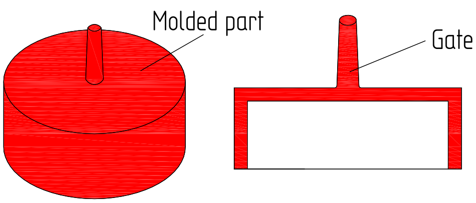

Figure 3.15 Principle of a direct gating

Table 3.4 Standard Values for Direct Gate Diameter

Weight molded part in g	Ø Direct gate in mm
0,5 – 10	2.5 – 3.5
10 – 20	3.5 – 4.5
20 – 40	4.0 – 5.0
40 – 150	4.5 – 6.0
150 – 300	4.5 – 7.5
300 – 500	5.0 – 8.0
500 – 1000	5.5 – 8.5
1000 – 5000	6.0 – 10.0

The diameter of the direct gate should not be much greater than the greatest wall thickness of the molded part.

3.2.1.2 Pinpoint Gate

The pinpoint gate is the simplest type of connection. It is particularly suitable for short cycles and very thin wall thicknesses of the molded parts.

Central pinpoint

The central pinpoint gate is carried out with a relatively large pre-chamber and a small gate suppository. It should not exceed 0.8 – 1.0 mm in diameter and height. The size of the pre-chamber design is crucial for successful injection molding of the parts. In an insufficiently dimensioned (too small) pre-chamber, there is a risk of the melt solidifying. The central pinpoint gate is particularly suitable for short cycles of 3 – 4 shot/min. It can only be used for simply designed molds.

Typical applications include cups, buckets, bowls, boxes, and packing materials.

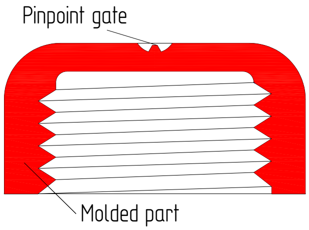

Figure 3.16 Principle of a central pinpoint gate

Lateral pinpoint

The lateral pinpoint gate is preferably used for molds with multiple cavities. The gate often remains on the injection molded part, and rework is omitted.

This type of connection is often chosen for molded parts that could get stuck at the gate and are sold as a set. Typical applications include model kits and parts that are further processed in assembly machines.

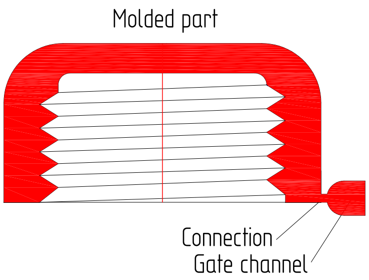

Figure 3.17 Principle of a lateral pinpoint gate

In the pinpoint gates, the distribution channel should be designed with a relatively large cross section until shortly before the mold cavities, which then leads into the short, narrow sprue channel that is a few tenths of a millimeter wide.

3.2.1.3 Fan and Ring Gate

For the injection of annular parts with unilateral core storage, the fan gate is used. The ring gate is used for annular or tubular parts with bilateral nuclear storage.

Fan Gate

The fan gate is mostly used for molds with a mold cavity. One condition for the application is a bore hole in the center of the part. Then, the sprue provides the optimal filling of the mold.

With the fan gate, round parts are produced with high concentricity without warping.

This has the advantage that weld lines are avoided, which are otherwise inevitably created during the molding of an annular cross section through several points. The gate may be separated in the bore hole without visible damage to the parts.

The most well-known application is the use in the injection molding of DVDs and CDs.

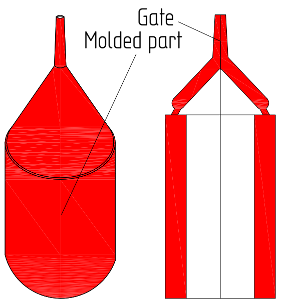

Figure 3.18 Fan gate

Ring Gate

With the ring gate, mainly sleeve-shaped parts are injection molded, in which the core has to be embedded due to its length. The ring gate prevents the displacement or bending of the relatively slender mold cores.

Typical applications for this are cartridges.

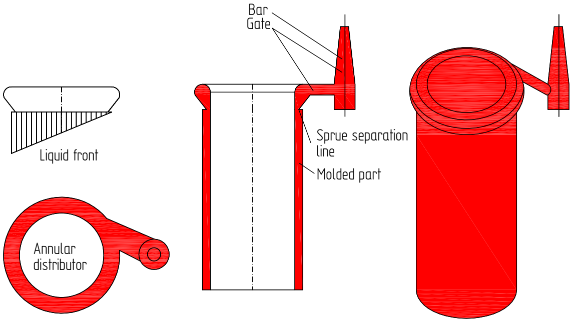

Figure 3.19 Ring gate

Both gate variants are particularly suitable for filling the molded parts so that the flow is uniform across the entire part.

3.2.1.4 Sprue Gate

The sprue gate is used for multi-cavity molds that do not make great demands on the surface of the molded parts. The molded parts are usually used with the resulting cavity without further rework.

The sprue gate is an advancement of the direct gate.

The sprue gate can be considered as physical distance to the gate system. Therefore, a thermal separation between the injection molded part and the hot runner system is reached.

The sprue gate offers, among all types of gates, the least resistance for the incoming mass. Through the tapered opening towards the part, the incoming mass relaxes. This requires a long holding time, but a low holding pressure.

The sprue gate is often used for thick-walled parts and for molded parts with long mold cores, or parts that are molded in a bore hole.

Typical applications include silicone cartridges. The sprue gate remains on the part and is cut off by the user.

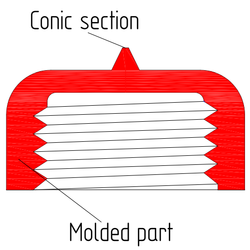

Figure 3.20 Sprue gate

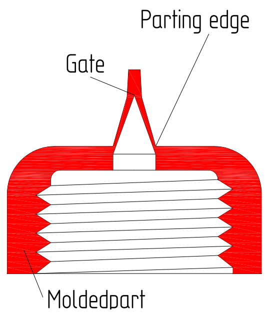

Figure 3.21 Sprue gate that can be cut off

The sprue gate is preferably used for thick-walled parts, as well as for molds with long cores that have to be centered during the injection phase. Injection and holding pressures remain effective for a long time.

3.2.1.5 Film Gate

The film gate is preferably used for large-area, thin-walled parts where no weld lines are permitted.

The film gate supports the uniform distribution of the plastic mass. It has nearly uniform shrinkage and low warpage in the flow as well as transverse direction. Unsightly gate marks on the surface do not exist.

Through the flat film gate, a one-sided load of the closing force is possible. This disadvantage can be avoided by using a dual mold design.

The mechanical removal of the gates can be done in many different ways depending on the application of the parts. Types of coarse separation include sawing, drilling, milling, turning, cutting, and punching. A dust-free separation form is done by electrically heated knife, laser, or water jet.

Film gates are especially used for flat injection molded parts, such as plates or strips, which may only show slight warpage and stress. Typical applications include household goods, industrial parts, rulers, and blanks for lens manufacturing.

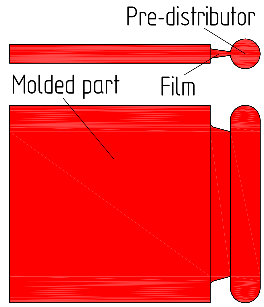

Figure 3.22 Film gate

Figure 3.23 Injection molded part with film gate (Source: FOBOHA)

Film gates are used whenever warpage caused by unequal mold filling is expected. Long flow paths inside the cavity of the film gate are reliably achieved.

3.2.2 Automatically Separated Gate

3.2.2.1 Tunnel Gate

This system includes gate systems that mechanically separate from the molded part when opening the mold. This means that molded parts and gates that are no longer connected to one another emerge from the mold.

The tunnel gate is so called because it cuts the molded part through a tunnel-like tube. The tunnel gate is an old but very popular gating system. It is often used because the manufacture is cost-effective.

The distributor channel leads into an inclined bore hole just before the cavity, which leads tunnel-like to a side-wall surface of the mold cooling. Depending on the design of the tunnel, the sprue (transition into the part) may be oval or crescent. If the funnel of the tunnel leads in the part with the full final diameter, the transition is oval-shaped. If the tunnel was created with a so-called conical bucket gate, the transition is crescent-shaped.

The better option is the design with the conical bucket gate, because the plastic melt, which is flowing ahead in the distribution channel, cools at the flow front. This cold plug is trapped in the conical bucket gate. From the plastic core of the distribution channel, hot plastic melt is now pressed into the mold cavity.

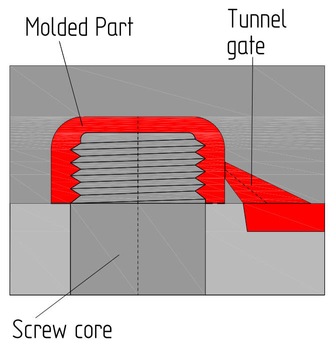

Figure 3.24 Tunnel gate

The tunnel gate is mainly used in multi-cavity molds for the manufacture of small parts. It is the only self-separating gating system with only one parting plane.

3.2.2.2 Whip Gate

Whip gates are used for molded parts that do not allow a gate mark on the visible surface, because they are to be either imprinted or chrome plated.

Whip or film gates are only used when the injection point should not be visible. Some parts are molded on the inside, which means through the mold core. This is an expensive technical mold solution.

Only tough, flexible plastics can be used for whip gates. They may not be reinforced with mineral or glass fiber.

An ejector pin should be attached close to the whip, which is lifting the gate, so that the whip can unscrew during ejection.

Since the whip gate cannot always tear without leaving a rest, the cutting location should appear to be deepened in the molded part to exclude a gate protrusion.

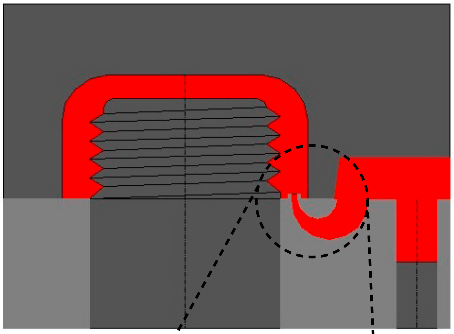

Figure 3.26 Principle of a whip gate screw cap

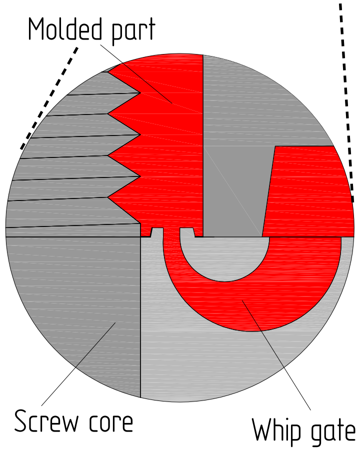

Figure 3.27 Principle of a whip gate (view: magnifying glass)

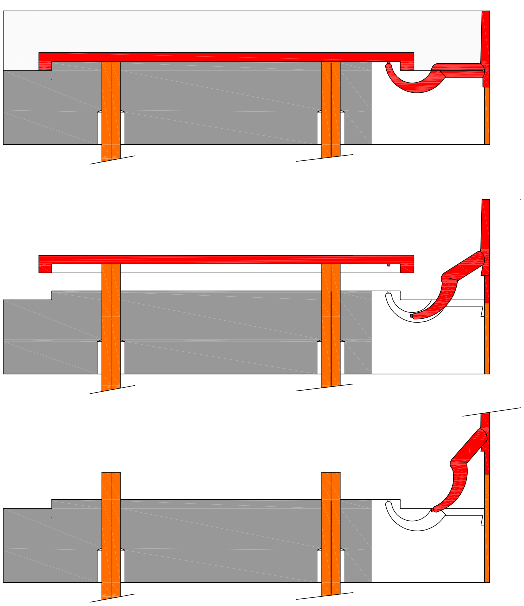

Figure 3.25 Ejector pins with whip gates

The whip gate is used when parts in the mold are separated from the gate system. Since the material is introduced into the wall from below, the gate mark is not visible.

3.2.2.3 Hot Edge Gate (Lateral Injection)

The "hot edge" nozzle offers new possible applications for hot runner needle shut-off nozzle systems. This allows a lateral injection of small parts with a needle shut-off nozzle system.

With the hot edge nozzle, molded parts can be laterally injected so that they can be ejected from the mold without a gate. In this case, there is no gate, because the nozzle is part of a hot runner system that is built in the mold.

The injection points of the molded parts are arranged on a reference circle or periphery of a circle. Parts that are arranged in series cannot be injected laterally. The hot edge nozzle replaces the tunnel gate. It improves the filling behavior by directly injecting at a right angle to the part.

This technique is very often used in the manufacture of small screw caps and tube caps. The molds are often occupied with up to 128 mold inserts. In this case, instead of 128 hot runner nozzles, only 16 eight-fold hot edge nozzles are required. This results in tremendous savings in terms of cost and space.

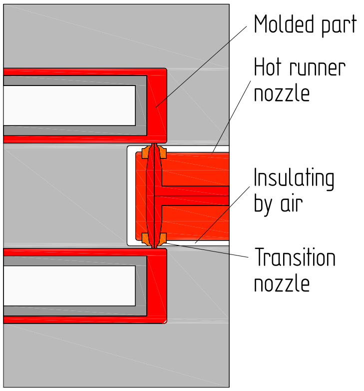

Figure 3.28 Principle of the hot edge nozzle

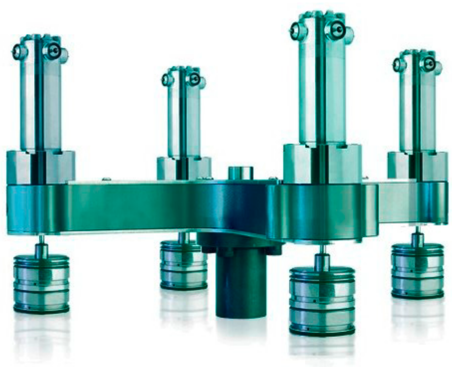

Figure 3.29 Side gate (Source: Otto Männer)

Through multiple nozzles with up to eight injection points, the distributor spider and thus the entire gate material is eliminated due to the hot edge nozzle.

3.2.2.4 Hot Runner Nozzles

In hot runner nozzles, a distinction is made between three different design principles: the open hot runner nozzle, the needle shut-off nozzle, and the internally heated hot runner nozzle, also called the torpedo nozzle.

Open and internally heated hot runner nozzles belong to the group of gating systems that are mechanically torn off when opening the mold. The sprues of the nozzles are formed as pinpoint gates. A variety of different hot runner nozzles is available today. The spectrum ranges from the nozzle head diameters of 10 – 60 mm and lengths of 50 – 300 mm.

The hot runner nozzles can be regarded as particularly critical components of the hot runner system. They have to meet the following requirements:

- uniform temperature control,
- thermal separation between hot nozzle and colder mold, and
- sealing of transition areas.

Internally heated hot runner nozzles are rarely used today because the temperature of the heating cartridge cannot be adequately controlled.

Needle shut-off nozzles are not among the nozzles that tear off when opening the mold. The separation only takes place when the needle in the nozzle closes. On the molded part, only an imprint, similar to an ejector pin, can be seen. For this reason, needle shut-off nozzles are often used for high quality plastic parts.

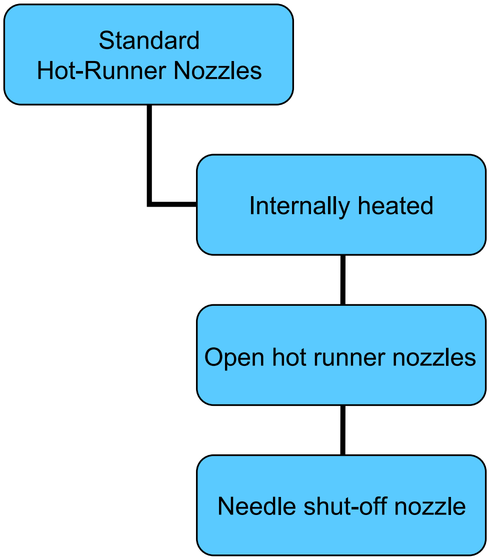

Table 3.5 Nozzle Variations

For different fields of application, there is a variety of hot runners for special processing of different materials.

3.3 Distribution System for Thermosets and Elastomers

Apart from some minor restrictions, the distribution systems of thermosets and elastomers are similar to the ones of thermoplastic distribution systems.

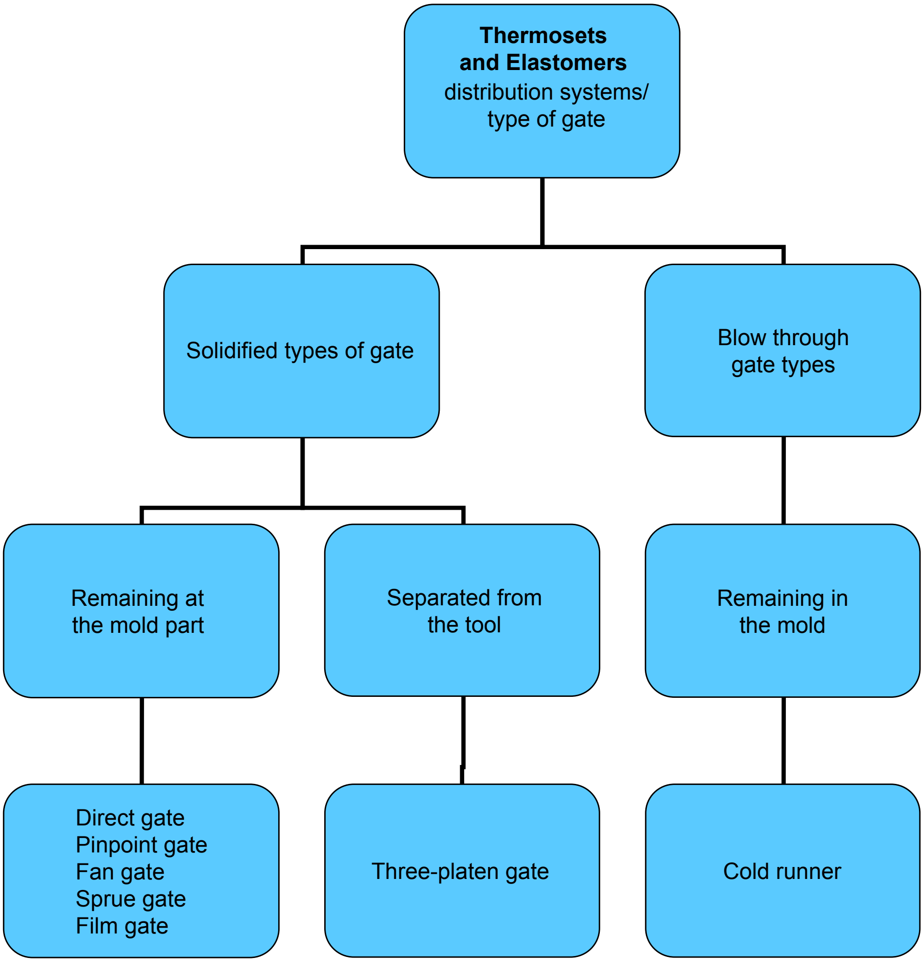

Table 3.6 Overview of Types of Gates/Distribution Systems for Thermosets and Elastomers

3.3.1 Demolding with the Molded Part for Thermosets

3.3.1.1 Cold Runner for Thermosets

In thermoset processing, only a low amount of thermal energy is introduced to the cold runner. Targeted tempering prevents heating of the distribution block. The distribution block is internally heated with through holes. Water is used as a medium because it reacts faster than oil.

In thermoset processing, a central direct gate is common. It is in regards to manufacturing the easiest, but not always the best solution. Many parts have a tendency to crack around the central direct gate. Thus, with regard to the temperature distribution and the homogeneity of the mass, the film gate is more appropriate. The film thickness should be between 0.5 mm and 0.8 mm.

The pinpoint gate is also often used because it can be separated from the part without rework. In multi-cavity molds, it is important that the flow paths are equally long and balanced. Unequal flow paths show an unequal flow pattern. A flow simulation shows where the gate should be adjusted.

The high filler content causes strong orientations on the surface. This can lead to warpage of the parts in an unfavorable situation position of the gate. The injection orientation should be in the direction of the main load. This increases the strength of the molded part.

Apart from a few exceptions, all types of gates can be used in the processing of thermoplastics.

The cold runner nozzles are externally heated with a circumferential circular jacket. For the nozzles, it has to be ensured that the contact surfaces are as small as possible. To control the temperature distribution optimally, only parallel cooling circuits can be operated.

In the cold runner and injection cylinder, the mass must be heated to 40 – 90 °C, which means that it is relatively cold compared to a hot runner. At production stop or extended interruption, the cold channel should be rinsed with a polypropylene to prevent the mass from solidifying in the cold runner.

3.3.2 Demolding with the Molded Part for Elastomers

3.3.2.1 Cold Runner for Elastomers

In the design of the cold runner, it is important that all cavities are balanced to the central injection point (i. e., are at the same distance).

The cold runner in elastomer processing is the counterpart to the hot runner in the processing of thermoplastics.

Cold runners are usually tempered with hot water or oil. The temperatures are between 40 and 90 °C, depending on the material type. It is important that the cold channel is shielded against the very hot mold so that no cross-linking takes place in the distribution bore holes of the cold runner.

The cold runner transfers the elastomer compound directly to the mold cavities. This area has to be tempered low until the transition into the mold cavity so that no solidifying of the mass can occur. The insulating channel is simply an extension of the injection unit up to the mold cavity.

With the VARIO cold runner technology by DESMA, nozzle spacing can be variably adjusted for the first time to cover different molds. The nozzle spacings of 240 – 460 mm are continuously adjustable, based on details provided by the manufacturer without changing the balancing.

Fully developed cold runner systems are offered by ordinary manufacturers of standard parts. A do-it-yourself version is hardly worth it.

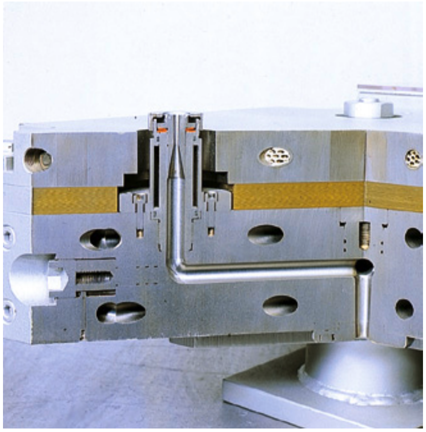

Figure 3.30 Standard cold runner (Source: Klöckner DESMA Elastomertechnik)

Figure 3.31 VARIO cold runner (Source: Klöckner DESMA Elastomertechnik)

In the elastomer molds, a cold runner introduces the elastomer mixture to the mold cavities. The temperature in the cold runner should be selected low enough that no vulcanization of the mass can occur.

References

1. Information publication Desma Elastomertechnik, Fridingen
2. Mennig, G., Wippenbeck, P., *Mold-Making Handbook* (2013) Hanser Publishers, Munich
3. Johannaber, F., Michaeli, W., *Handbuch Spritzgießen* (2004) Hanser Publishers, Munich

4 Standard Parts

4.1 Mold Designs

The use of prefabricated, standardized assemblies or components facilitates the work of the mold maker considerably. The standard mold consists of the lower part with the guide columns and the upper part.

Originally, the plate sets only consisted of ejector- and nozzle-sided guide columns, but complete mold assemblies are now available.

The mold maker can completely concentrate on the molded parts, such as cavities. The standard parts dramatically reduce both the vertical range of manufacture and the processing times.

Standard mold assemblies are now available in various sizes. They range up to 796 × 996 mm with plate thicknesses that vary from 10 to 196 mm. The quality of the steel can also be chosen differently.

All plates are sanded on each side and are carried out with a plane parallelism of 0.008 to 100. The thickness tolerance is +0.05 – 0.25 mm. With this design, the mold makers can process the plates without rework. The required guide elements such as columns, bushings, and sleeves are adapted to each plate size.

A wide range of hot runner technology and nozzle series up to melt-flow and temperature-control circuits completes the program.

Figure 4.1 Standard parts for standard mold (Source: HASCO)

Standard parts reduce both the vertical range of manufacture and the processing times for the manufacture of a mold.

4.2 Mold Guide Elements

Guide elements are necessary to avoid damaging the mold inserts and cores when closing the mold. For small and light molds, guide pins or bushings are usually used.

Mold guide elements are hardened and can be replaced in case of wear. To prevent the plastic parts from being smeared with grease, the guides usually run dry (i.e., without lubrication). The standard parts manufacturers offer self-lubricating, maintenance-free guide bushings. The use of oils and grease is unnecessary. These guide elements can be used at mold temperatures of up to 200°C.

Depending on the requirements, round or flat centering units are available. Special centering elements relieve the stress on the guide members in a closed mold and achieve higher accuracy in a tight fit.

In medium- to large-sized molds, it is recommended that the molds be supported on the injection molding machine. With a support element (the brace), the mold is supported on the bars of the injection molding machine or allowed to run along the platen guide of the machine.

Figure 4.2 Guide pin and bronze bushing with graphite lubricant depots (Source: HASCO)

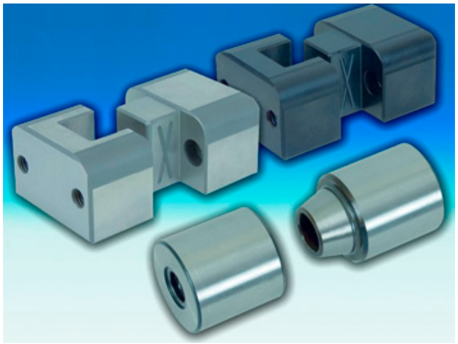

Figure 4.3 Flat and pin centering (Source: HASCO)

The most important guide elements are centering sleeves, guide pins, and guide sleeves. The standard parts manufacturers offer guide elements that are almost exclusively standardized according to DIN/ISO.

4.3 Demolding Elements

The demolding elements include ejector pins, collapsible cores, sliders, jaws, and latch locks in all variations.

Ejector Pins

Ejector pins must be selected so that the diameter is large enough that the demolding force is sufficient to eject the part. The ejector force must be greater than the adhesive force of the molded part in the mold cavity. If the adhesive force is too large, deformation or destruction of the molded parts often occur during demolding. The placement of the ejector pins must be arranged in such a way that the demolding force does not exclusively affect the size of the molded parts.

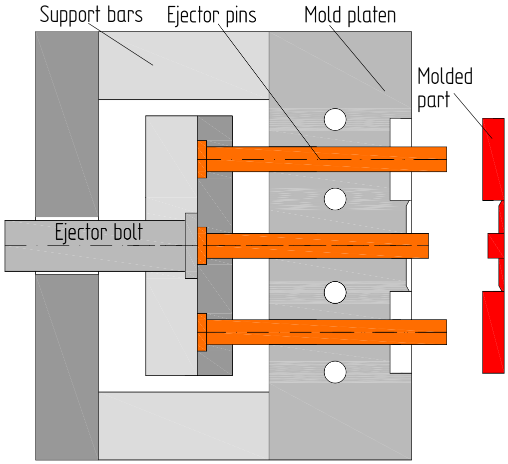

Figure 4.4 Principle of ejector pins

Collapsible Cores

For demolding parts with internal undercuts and internal threads, collapsible cores should be used. The cores are hardened and are, except for the required part thread, ready for installation.

Figure 4.5 Collapsible core (Source: HASCO)

Sliders, Jaws

External undercuts are usually demolded by means of laterally moving elements; these include sliders, jaws, and core pullers.

Latch Locks

In molds with multiple parting planes, latch conveyors are required to open the second parting plane.

In visible parts, mold release agents must be used so that no visible marks are formed on the molded part.

4.4 Equipment for Mold Temperature Control

For effective mold cooling, a number of standard parts are available. They range from water/oil connection couplings via hoses, core pins, and deflection elements, up to spiral cores for cooling.

Water Connection Couplings

The standard parts manufacturers provide a wide range of water/oil connection couplings. The most common system diameters are 9, 13, and 19 mm. For the temperature control of very small cores, the nominal size is 5 mm.

Quick release couplings are offered with or without a check valve. While the open systems are characterized by a lower pressure drop in the cooling circuit, closed systems prevent the leakage of the cooling medium when releasing the couplings. This also prevents the penetration of oxygen, which leads to corrosion.

Figure 4.6 Quick-release coupling (Source: HASCO)

Spiral Cores

A spiral core is embedded into the core bore hole with a precise fit, which leads the temperature control medium perfectly along the bore wall and ensures good heat transfer. Spiral cores may be single- or double-threaded. In single-threaded spiral cores, the medium is guided into the core through a core hole and then flows back through the coil. The double-threaded spiral core guides the temperature-control medium through a helical turn in the core and through a second helical turn back to the outlet.

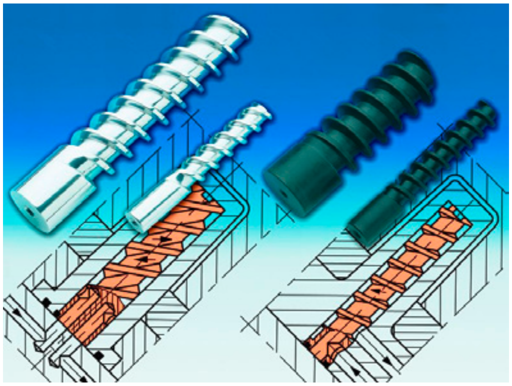

Figure 4.7 Spiral cores (Source: HASCO)

4.5 Mold Quick-Change Systems

To quickly change molds means not to just clamp quickly, but also to connect with all necessary units.

The clamping of the molds requires the least amount of time when changing a mold. The coupling to the machine ejector, the application of the cooling circuits, the connection of the limit switches and the mold heating, and connecting the core pullers to the injection molding machine require the largest amount of time when changing the mold. All of these components are important for a good mold change system.

A fully automated solution is only offered by about 0.5 % of all injection molding companies. A mold change can be partially automated with much lower costs. Many of these partially automated components are offered by standard parts manufacturers.

Individual steps of partial automation include:

- use of mechanical, hydraulically operated mold clamping elements or magnetic clamping plates,
- quick couplings for water and hydraulic oil,
- quick couplings for ejector systems, and
- plug-in system for hot runner and limit switches.

The following practices have been proven effective:

- The connections for the media should be placed on supply blocks, which are screwed onto the clamping platens of the machine.
- The cooling connections for flow and return should be marked in color.
- The plug elements for the electrical supply should be placed on the mold such that no confusion occurs.

Taking small, intermediate steps towards partial automation are helpful, cost-effective, and save time when changing molds. These steps also eliminate error.

4.6 Latch Conveyors

Latch conveyors allow the mechanical movement of plates during the mold's opening movement.

The additional movement of a plate or the fixing of a stripper plate are typical tasks that can be solved by a latch conveyor. These functions, which create an additional parting plane, are only performed during the plate's upward motion. During the opening movement, the latch conveyor takes a stripper along until it is moved back through a mechanical control slope. and thus the stripper plate comes to a stop, although the mold drives up further. The latch conveyor must not be monitored by any electrical control because the latch releases mechanically. Because these elements are used so often, all standard parts manufacturers offer various latch conveyors.

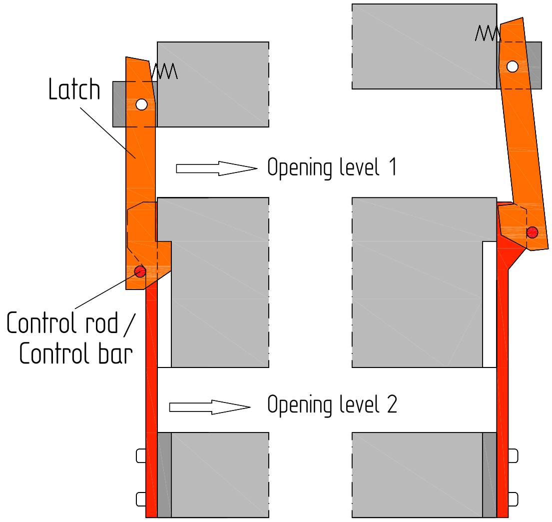

Figure 4.8 Principle of a latch conveyor

In the latch conveyor, the functions are purely mechanically force-guided. There is no need for electrical activation or function monitoring.

4.7 Hot Plates for Thermosets and Elastomers

In thermoset or elastomer processing, injection molding machines are often equipped with hot plates. This has the advantage that components for heating the mold need not be installed for every mold.

Heating plates are fixed to the nozzle or ejector side of the injection molding machine. They are equipped with heating elements and sensors necessary for controlling the temperature. The heating plates with heating cartridges or heating coils are equipped so that different temperature profiles can be set. This is important in order to cover the heat requirements of the various molds.

Hot plates have a heat output of about 5000–7000 watts. They cannot be used universally. They are preferably used for simple, flat parts. In molded parts that have strong height differences, hot plates are not sufficient because the temperature differences would be too large within the mold. In such cases, it is better to operate the mold with a separate heater.

By skillfully placing heating elements and temperature sensors, the temperatures at the mold surface can be effectively controlled. The temperature differences should be ±3°C over the entire mold surface. As a result, a uniform hardening is achieved, which prevents internal stresses from building up and warping the injection molded parts.

4.8 Brushing Units for Elastomers

Many elastomeric parts are flexible and unstable. They cannot be ejected with normal ejector elements. These parts must either be removed from the mold by hand, with a brushing unit, or with a robotic system.

Brushing units slide over the mold surface with rotating brushes that brush the parts out of the mold cavities with the counter-rotating movement of the brushes. These units offer a variant for the ejection. With these brushes, the mold flash, distribution systems, and the distribution channels are simultaneously brushed out of the mold. Double brushes are often used to ensure that no parts or residue of the distribution system remain on the ejector and nozzle side.

The brushes are mostly made from natural bristles, either leather or rubber stripping. They have to withstand temperatures of up to 250 °C. Therefore, not all materials can be used.

It is also important to be able to vary the brush diameter by replacing the brush head. The brushing units can also control whether the brush touches aggressively or softy.

For 100 % control of ensuring a clean mold, image surveillance systems are available. These systems detect when the gates or parts get stuck in the mold and initiate a second or third cleaning step.

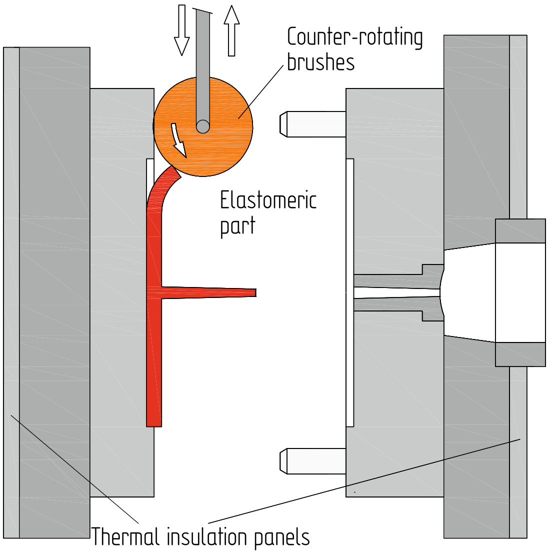

Figure 4.9 Principle of brushing elastomer parts

References

1. Mennig, G., Gockel, J., Kabbert, A., *Mold-Making Handbook (2013)* Hanser Publishers, Munich
2. Mennig, G., Hentrich, R., *Mold-Making Handbook* (2013) Hanser Publishers, Munich
3. Mennig, G., Karlinger, P., Hinken, F., *Mold-Making Handbook (2013)* Hanser Publishers, Munich

When brushing, the parts, the mold flash, and the distribution systems are brushed out of the mold.

5 Temperature Control

5.1 Temperature Control Channels

Temperature control channels have the task of transferring heat from the mold to the temperature control medium through their lateral surface. The most cost-effective type of temperature control uses cooling channels with round geometries.

Each molded part must be individually temperature controlled. There are general rules for determining the temperature control channels:

- The channel diameters are usually 6 – 14 mm.
- Multiple temperature control channels with a smaller diameter are better than a few channels with larger diameters.
- Long channels with small diameters lead to large pressure losses.

Series Temperature Control

In the series temperature control method, only one channel with an inlet and outlet leads through the mold. One consequence of this method is that the temperature of the medium increases through the course of the channel. Thus, different mold regions have different mold temperatures, which is a disadvantage.

Parallel Temperature Control

In this temperature control method, a feeding channel is divided into multiple parallel channels to achieve a uniform mold temperature control. The disadvantage of this method is that part blockages are detected only with great difficulty.

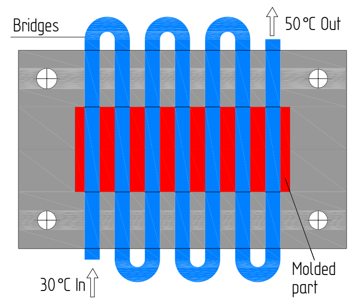

Figure 5.1 Series temperature control

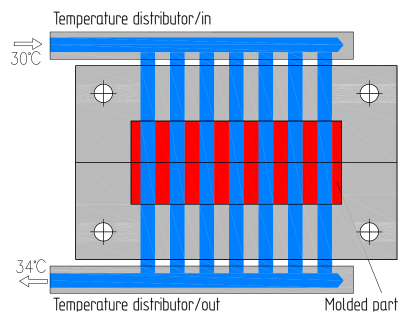

Figure 5.2 Parallel temperature control

In general, temperature control channels are located close to the mold surface. The distance of the channels should be as small as possible; the heat exchange surface should be as large as possible.

5.2 Temperature Distribution

The mold temperature is essential for the economic efficiency of injection molding and for the quality of the molded parts. According to the latest research, the temperature deviation over all the cavities should not exceed 5 K. A series of cooling measures allows this.

In the past, the temperature difference between inlet and outlet was often more than 20 K. This resulted in degradation in quality of the molded part and long cooling times of the solidifying melt. The cycle time was correspondingly extended. Today, the mold temperature control is designed so that heating is done selectively, that is, close to the surface and divided into zones.

There are various methods of mold temperature control that allow the demolding temperature to be reached faster. Briefly, these are the continuous and the segmented temperature control, as well as the discontinuous temperature control, also called pulsed cooling.

In all cases it is important to remove the heat at the proper temperature level. Rapid cooling with low mold temperatures has a negative influence on the part properties.

In the temperature distribution, the flowability of the plastic material should also be considered.

The ideal mold temperature for the different plastic types is based on recommendations of the material manufacturers and shown in Table 5.1.

Table 5.1 Temperatures of Common Plastic Materials

Plastic Material	Melt Temperature °C	Mold Temperature °C
ABS = Styrene Copolymer	210 – 270	50 – 85
PE = Polyethylene	180 – 270	20 – 60
PMMA = Polymethylmethacrylate	200 – 250	40 – 60
PP = Polypropylene	240 – 300	20 – 80
PS = Polystyrene	180 – 280	55 – 80
PVC = Polyvinyl chloride	170 – 190	40 – 60
PA = Polyamide	230 – 250	70 – 120

The mold temperature control has a decisive influence on the quality of the molded parts and the cycle time of the process. To produce perfect parts, the same temperature should prevail in all areas of the mold surface.

5.2.1 Temperature Distribution in Thermosets

Thermoset molds are electrically heated. This is done by heating cartridges, heater bands, heating coils, or hot plates. Hot plates are not integrated into the mold, but are part of the injection molding machine. The mold temperature is between 150 and 180 °C.

Temperatures are measured with thermocouples. The thermocouples and the heating elements have to be placed so that surface temperatures are within 5 K. At very high requirements on the molded part, it is possible that the required tolerances are as low as 2 K.

To keep the radiated heat of the molds as low as possible, the molds must be equipped with insulation plates to prevent heat loss.

5.2.2 Temperature Distribution in Elastomers

Elastomer molds are electrically heated. This is done by cartridge heaters, heater bands, heating coils, or hot plates. Heating plates are usually not integrated into the mold, but are part of the injection molding machine. While the heat output in thermoset molds is about 35 – 40 Watt/kg, 50 – 60 Watt/kg are needed for elastomer molds. The reasons for this are the low thermal conductivity of the elastomers and secondly the faster cooling of the elastomer molds due to their weaker and thinner design.

Temperatures are also measured with thermocouples. The thermocouple and the heating elements have to be placed so that surface temperatures are within a tolerance of ± 5 K. At very high requirements on the molded part, it is possible that the required tolerances are as low as 2 K.

To keep the radiated heat of the molds as low as possible, the molds must be equipped with insulation plates to prevent heat loss.

Elastomer and thermoset molds have no cooling system, but a heating system. The injection is done at a mold temperature of 150 to 180 °C. To produce perfect parts, the same temperature should prevail in all areas of the mold surface.

5.3 Continuous Cooling

A cooling medium can be used as a mold temperature control, and it can be directly led through the mold. A majority of injection molds, however, are heated to a temperature between 40 and 80 °C by a temperature control unit. Engineering thermoplastics require mold temperatures of up to 200 °C.

Cooling Water Flow Controller

Cooling water flow controllers have a manual volume control with temperature control. A thermometer measures the temperature in the return and responds to temperature changes. Adjustment of the flow rate is possible with control valves.

Displaying of the flow rate takes place after the heavy body measuring system. That means a cone is lifted from the flowing water in the return of the sphere.

Temperature Control Units

In the temperature control unit, the temperature media used is water or oil. If temperatures higher than the boiling point of water are required, pressurized water or oil systems are used.

The circuits in conventional temperature control units can be operated either open or closed. Temperature control units in the open-construction method and indirect cooling can be used in the forward motion to about 95 °C for water and up to 200 °C for oil. Oil is usually used in a closed circuit in order to avoid contact with oxygen. In closed circuits, temperatures of up to 230 °C are reached in the water units.

If the temperature control medium does not reach directly into all areas of the shaping components, the cooling time must be extended to allow the demolding temperature to also be reached in the noncooled areas. This means long cycle times and therefore higher parts cost. Therefore, all the components of the mold must be integrated into the cooling circuits.

Figure 5.3 Cooling water flow controller with 6 circuits (Source: Wittmann Kunststoffgeräte)

Figure 5.4 Temperature control units (Source: gwk Gesellschaft Wärme Kältetechnik)

In direct cooling, the water of the in-house network is directly routed through the mold. In indirect cooling, temperature control units equipped with heat exchangers are interconnected. The pump should guarantee a difference of less than 3 K between the inlet and outlet temperatures in the mold.

5.4 Segmented Temperature Control

The basis of segmented temperature control is the thermal mold design based on parts segments. The mold is divided into individual segments in order to assign different temperature requirements to individual temperature circuits.

In the conventional technical solution of the segmented temperature control, also called multiple-circuit temperature control, individual temperature control units may be connected for the different circuits. As the number of individual temperature control circuits rises, much effort is necessary and space requirements become high.

Depending on the molded part, segmented temperature control allows it to cool individually. In long flow paths, intensive cooling can be done close to the gate and the temperature can be slowly reduced further away from the gate. The various temperature zones lead to a uniform cavity temperature. Thus, the molded part quality is improved and the cycle time is reduced. Temperature control circuits at the same temperature can be compiled, which reduces the temperature control effort.

Cooling will be difficult especially for bars and thin cores. A good compromise for these areas is to use materials with higher thermal conductivity. For example, copper beryllium alloys conduct heat five times better than steel.

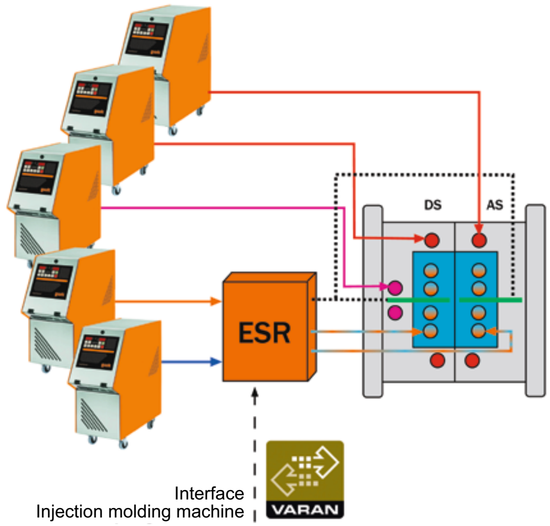

Figure 5.5 Segmented temperature control (Source: gwk Gesellschaft Wärme Kältetechnik)

Depending on the molded part, segmented temperature control allows one to connect different circuits to each separated, single-circuit temperature control unit. Thus, individual zones can be individually temperature controlled depending on the supplied heat.

5.5 Dynamic Temperature Control

Dynamic temperature control is referred to as a variotherm process control. Here, the forming mold cavity is brought (cycle dependent) to different temperature levels by alternate heating and cooling.

This method works on the principle of securing the flowability of the molding compound by actively increasing the mold wall temperature before injection and up until the mold is completely filled. A better impression of the surface is thus achieved at low pressures.

For the external process, the heating is carried out via a liquid or vaporous medium, alternatively via induction or infrared radiators, which only heat a thin layer of the cavity. The internal processes work with electrical heaters, which are integrated to the mold close to the cavity. These may be heating cartridges, induction loops, or ceramic resistance heaters. The cooling is in general discontinuously carried out with water.

In the classical variotherm temperature control, the two-stage temperature control is achieved so that the cooling channels are first perfused by water at high temperature from a temperature control unit. Then, there is an exchange with a colder temperature control circuit.

A further development of the process replaces the fluid heating circuit with a high-performance ceramic heater located a few millimeters behind the cavity, which allows the desired temperature change to be induced up to ten times as fast with only one-tenth of the energy expenditure.

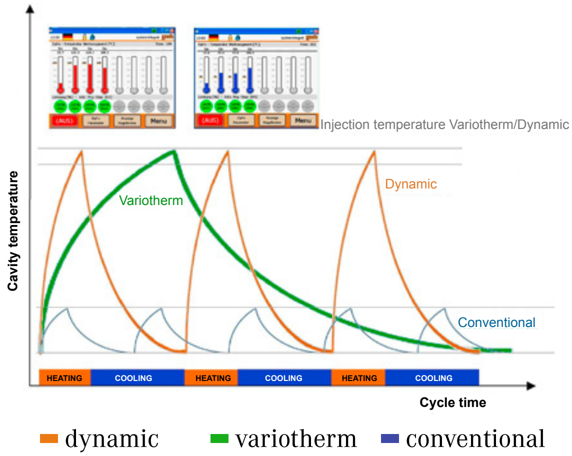

Figure 5.6 Dynamic temperature control (Source: gwk Gesellschaft Wärme Kältetechnik)

Variothermal processes enable the production of weld-line-free surfaces.

5.6 Pulsed Cooling

With pulsed cooling, a targeted mold surface temperature distribution can be created in the mold cavity at any time of the injection phase.

In this system, the cyclical progress of the cavity temperature is detected by a thermocouple and is used for the temperature control. A continuous monitoring of heat development during the injection phase is possible through thermocouples and flow sensors. If necessary, a cooling medium can flow (computer-aided) through the mold zones.

The heat balances are created by measuring the amount of cooling water and the temperature difference in each cooling channel of the mold. The individual cooling demand is controlled by the controller via the valve opening times for individual cooling circuits.

At the beginning of each production, the system to be temperature controlled should first be brought to process temperature. Alternatively, in some cycles, it is necessary to wait until the supplied heat reaches the required operating temperature. Only then, can the pulsed cooling start.

According to the manufacturer specifications, pulsed cooling reduces the cycle time by up to 30 % and reduces energy costs for cooling and temperature control of the molds significantly.

Typical applications include injection molded parts with large wall thickness variations. These are especially used for technical parts with ribs or with so-called “potspots.”

Pulsed cooling can realize, with only one flow temperature, different temperatures in the injection mold by cycle-dependent cooling pulses. The temperature control is perfectly adapted to the thermal profile of the molded part.

5.7 Core Temperature Control

Core temperature controls run mostly by a temperature control finger, which leads the temperature control medium into the core and then removes it again. In practice, there are a number of core temperature control possibilites.

Temperature Control Tube

In this variant, a temperature control tube is introduced into a core hole and supplied by a feed. The temperature control medium flows through the tube, escapes at the front side, and flows between the core hole and the temperature control tube back into the outlet.

Separating Plate

The separating plate is the simplest form of core temperature control. The core hole is divided into two chambers that are interconnected. The temperature medium can flow into the core hole via the feed. The cooling medium flows back into the outlet via the second chamber.

Heat Conductive Cartridges

Heat transfer cartridges are ideal for all applications where the temperature control channel has no direct access. Using the cartridge, heat can be locally removed and dissipated at a geometrically favorable position.

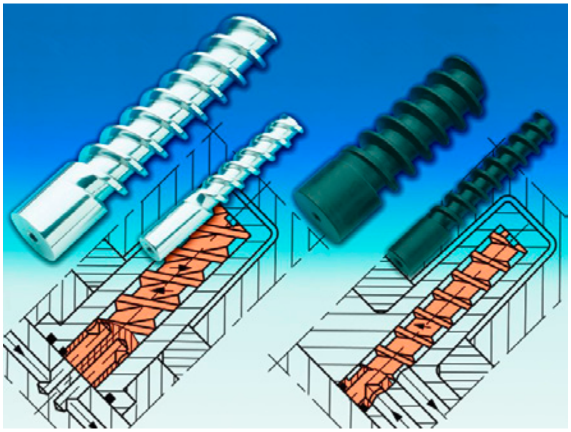

Figure 5.7 Spiral core (Source: HASCO)

The advantage of the core temperature control is the fact that the cooling liquid is distributed over a large area by means of cooling cores. Therefore, the heat transfer takes place rapidly. The closer the cooling circuits are to the mold wall to be cooled, the more intense the cooling performance.

5.8 Temperature Measurement

The temperature measurement in the mold is an important factor for the injection molding process. To obtain a process-oriented statement, the measurement of the temperature should take place in the gate area or in the mold cavity.

The measurement of the actual condition of the mold temperature is used as information to adjust the cooling or temperature control unit.

Any change in temperature in the injection molding process is detected early and allows a timely adjustment of the cooling medium. Thus, any process that could result in ruined materials is largely eliminated.

With a thermocouple, the mold temperature is recorded and reported either to the machine control or to the mold temperature control system (Mold Monitor). Both, through the machine control as well as with external temperature control devices, the set cavity temperature can be monitored. If the temperature deviates upwards and downwards from the predefined tolerance range, an alarm is triggered. The tolerance band is determined by the temperature sensitivity of the material to be processed.

The thermocouple should be located at a position in the mold where the temperature of the mold quality is crucial, such as areas with warpage tendency or closely toleranced dimensions. Cavity temperatures are often measured by sheathed thermo elements with protective pipe diameters from 1 mm. To keep the heat transfer resistance as small as possible, the use of thermal pastes is recommended.

Figure 5.8 Thermocouple (Source: Kistler Instrumente)

Thermocouples have a punctiform measuring point. The position of the thermocouple is dependent on the geometry and the construction of the mold, as well as the arrangement of the cooling channels. Thermocouples are preferably iron constants such as FeCo, Fe-CuNi, and NiCr-Ni.

References

1. Hoffmanns, W., “Impulse für das Temperieren,” *Plastverarbeiter* April (2004) Hüthig Publishers
2. Martin K., “Konzepte für die Werkzeugtemperierung,” *Kunststoffe* November (2002) Hanser Publishers, Munich
3. Gries, H., “Heiß und Kalt,” *Kunststoff Trends* March (2002)
4. Mennig, G., Thiemel, P., *Mold-Making Handbook* (2013) Hanser Publishers, Munich
5. Gries, H., “Mit High-Tech Anwendungen im globalen Wettbewerb erfolgreich,” *Kunststoffberater* September (2010) Giesel Publishers

6 Special Designs

6.1 Stack Molds

Stack molds simultaneously produce molded parts in two levels, while the parts fall out of only one level in single-parting plane molds. This means that the number of parts ejected is almost doubled.

In principle, a stack mold consists of two ejector sides and a centrally located package. This package includes the hot runner and the nozzle sides on the right and on the left. The hot runner is either supplied with a central sprue bushing, a so-called "snorkel," or with two opposing needle shut-off nozzles with melt from the injection unit.

The synchronization of the stroke movement of the center plate is taken over by racks. The parallel guide of the mold halves is done by a column guide system. Due to the weight of stack molds, the guidance and the support of the plates is of great importance. In large, very heavy molds, the central package is supported at the machine bed by means of additional support.

Stack molds are often modularly designed. This allows for a fast and efficient exchange of mold inserts for product families, such as containers of different sizes.

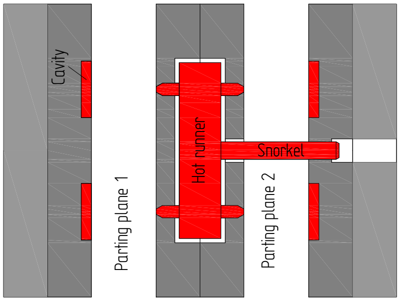

Figure 6.1 Principle of a stack mold

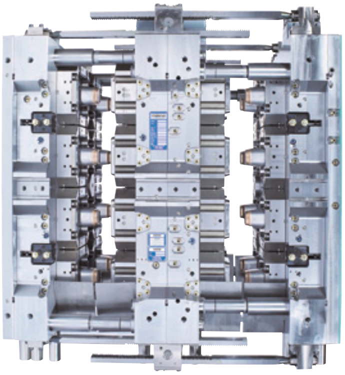

Figure 6.2 Stack mold (Source: Otto Männer)

Stack molds allow doubled output at the same clamping force. Their use only makes sense if large quantities have to be produced in a continuous operation.

6.1.1 Stack Molds in Elastomers

With the two-level, cold runner stack technology, production can be increased by more than 70 %, and the energy consumption is significantly reduced by not requiring a second injection molding machine. Manufacture with stack molds is mainly used with clamping forces ranging from 3,000 to 10,000 kN.

The cold runner distributor with cold runner nozzles must not be heated above about 60 °C. The elastomer material must lie below the vulcanization temperature. The heating of the mold plates, the mold inserts, and the cores is carried out electrically.

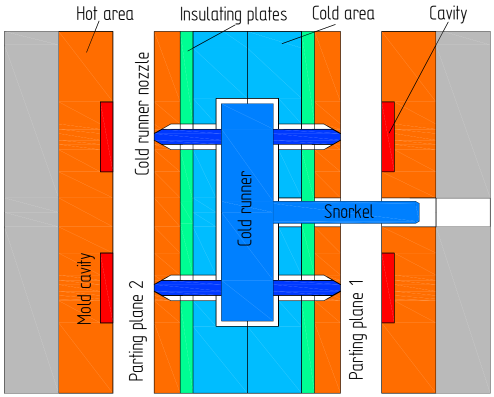

Figure 6.3 Principle of a stack mold with cold runner

Figure 6.4 Center distribution plate (Source: Klöckner DESMA Elastomertechnik)

With stack molds, production can be increased by around 70 %, with only 30 % higher investment costs.

6.2 Multi-Component Molds

In multi-component injection molding, injection-molded parts can be manufactured in different procedures using different plastics or multiple colors.

The multi-component technology has become more important in the last few years because the variety of applications is constantly increasing. These ever new and innovative solutions make multi-component injection molding attractive in an expanding market.

A major reason for the growth of multi-component technology is the potential savings achieved by reducing the production steps. Through skillful mold technology, manual or automatic assembly operations can be done in the mold. The design aspects, protection against plagiarism, and better haptic properties have also made multi-component technology attractive.

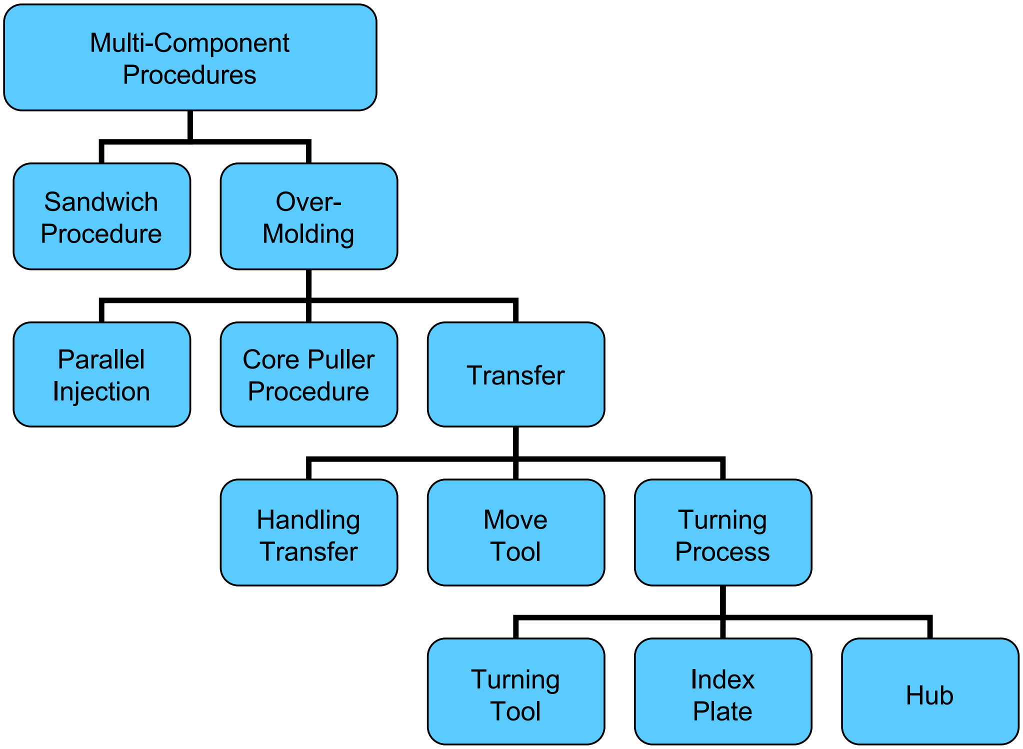

Table 6.1 Multi-Component Processes

6.2.1 Slider Technology (Core-Back)

The big advantage of this process is the flexibility to choose the point to inject the second component. Only after the retraction of a movable slider from the cavity of the preform, does the second component reach into the vacant portion of the molded part.

With the core-back method, it is possible to produce compatible material combinations in a fixed connection. Even before the first component is solidified, the second component is injected. This mold does not need to be moved or opened.

The connection between the two plastic elements is very homogeneous due to the fast sequence during injection. The actual geometrical hooking bondage is achieved by the second component. In the molten state, the plastic can easily fill breakthroughs in the geometry of the first component. This technique is very simple and space-saving.

Since no parallel injection is possible in the core-back process, injection times of the two material components add up. The sequential injection of the various components increases the overall cycle time. Because of this disadvantage, the core-back method is used increasingly less often.

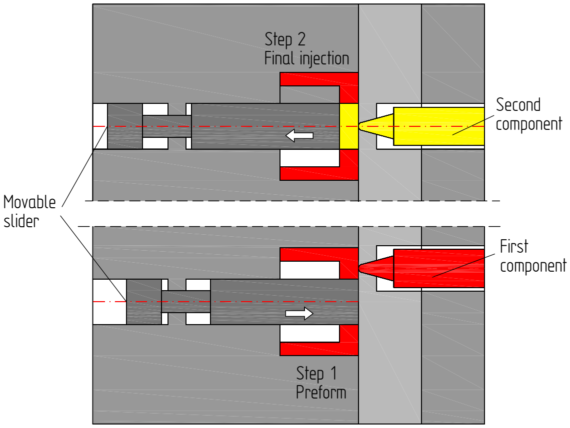

Figure 6.5 Principle of the core-back method

With the core-back method, incompatible material combinations can create a fixed connection in a very simple and time-saving manner. This cost-effective solution is preferably used for the production of small quantities because of its extended cycle time.

6.2.2 Transfer Process (Handling Transfer)

The transfer process is mainly used when molded parts have to be overmolded. Preliminary and final injection stations are located side-by-side or on top of each other. A handling device transfers the preforms and removes the finished parts.

This mold, like the one in the core-back process, is not very complicated because it works without any rotational movement. The advantage is that both materials are injected simultaneously, which significantly reduces the cycle time compared to the core-back method.

The transfer process also includes the injection of a part on one injection molding machine and the subsequent removal and insertion of the preform into a second machine for the injection of the second material.

The process-reliable insertion of the preform into the next cavity is critical. The empirical adjustment of the handling device is correspondingly complex. The exact centering of the preforms in the finishing station requires a precisely controllable gripper system.

The transfer process provides a good use of the available clamping plates. The transfer technique is not suitable for molded parts with filigree geometry.

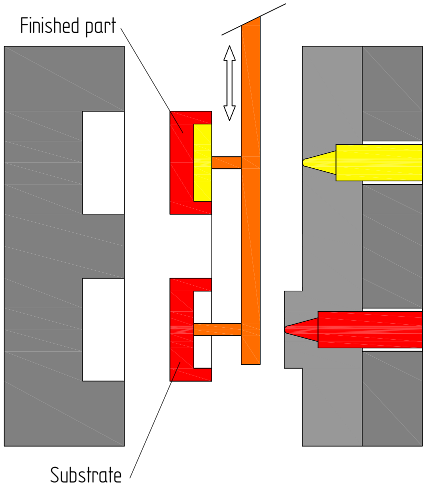

Figure 6.6 Principle of the transfer process

In the transfer process, no rotational movement of the mold is needed. The transfer takes place with external or internal handling equipment. Injecting is done simultaneously in all cavities.

6.2.3 Index Plate

The index plate is integrated into the moving mold plate and is pivoted. After opening the mold, the plate transports the preforms into the second station of the mold for final injection of the parts.

In index plate molds, an additional, rotating third plate is located between the two mold halves. It is rotatable about a central axis. The index plate is first released through the ejector function by the ejector-sided mold half and then rotated to the second station with a shear and rotating spindle. The drive of the rotary movement of the rack is carried out with a hydraulic motor or a servomotor positioned to within a few hundredths of a millimeter.

After the return of the lifting function, the positioning of the core insert in the movable mold half takes place again. Thereafter, the mold closes and the next injection operation begins. In the second station, the preforms can be overmolded with another component.

The rotation of the index plate may be 2 × 180° or 3 × 120°. The third station is often used for cooling and removal of the molded parts. The use of hot runner systems in index molds is possible only to a limited extent.

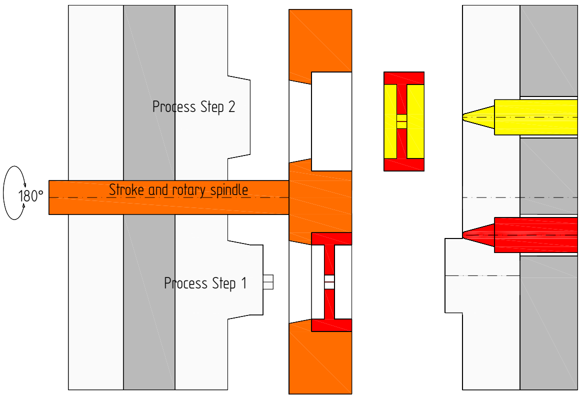

Figure 6.7 Principle Index plate mold

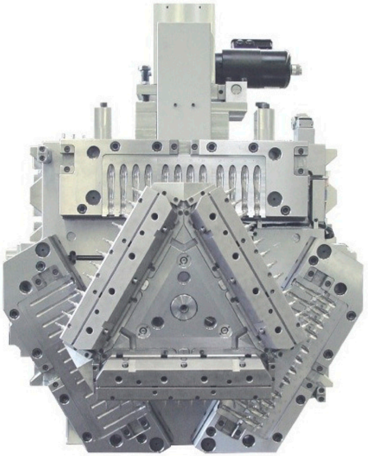

Figure 6.8 Index plate mold (Source: Zahoransky Group)

The transport of the preforms is done via a rotary core insert that is coupled to the ejector. The rotation of the index plate can be carried out in a continuously rotating motion at 120° or in a pivoting motion by swinging around 180° back and forth.

6.2.4 Hub

The hub system is similar to the index plate technology and is used very often. The index plate is reduced to a bar or a cross. The hub only rotates the molded parts, without any mechanical components, into the next station. The molded parts are held by collapsible cores, pins, or bolts during transfer.

The index plate is reduced to a hub, which further rotates the molded parts. The injection is carried out directly. A hot runner system is often used. One advantage over the index plate is the low weight of the rotating system. This enables a very fast rotation or turning, which reduces the cycle time significantly.

The shaping of the molded parts is carried out both on the injection and on the ejector side. The molded parts are only held on collapsible cores when transferring, which release a bore hole during ejecting.

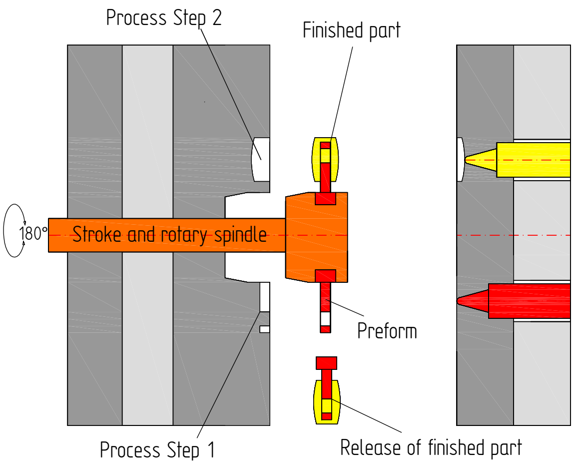

Figure 6.9 Principle of a hub

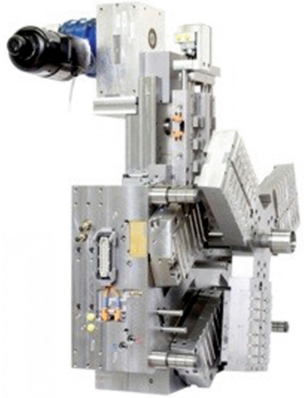

Figure 6.10 Hub mold (Source: Zahoransky Group)

In the hub, the molded parts are held by collapsible cores when transferring to the next injection station. The low weight of the hub allows a rapid rotation or turning movement.

6.2.5 Rotary disk

Multi-component molds with rotary disks are used in all areas of the plastics industry. Depending on the application, rotary disks can be driven hydraulically or with a servomotor.

The rotary disk is an economical solution for transporting the mold from one injection station to the next. The rotation of the mold is shifted to the rotary disk, whereby the injection mold becomes simpler.

Depending on the number of components, the rotary disk can be positioned in 4 × 90°, 3 × 120°, or 2 × 180° increments. In the simplest case, the rotary disk is pivoted by 180° to the left or right. The continuous rotation of the movable mold half is particularly useful for multi-station molds. The supply for cooling and hydraulics is accordingly complicated because the continuous rotation does not allow cable or hose connections.

A disadvantage of the rotary disk is the higher space requirement in the injection molding machine compared to other multi-component mold systems. Injection molding machines with rotary disks generally require a rail extension at a minimum of 200 mm. The distance between the rails is also increased by 50 – 100 mm.

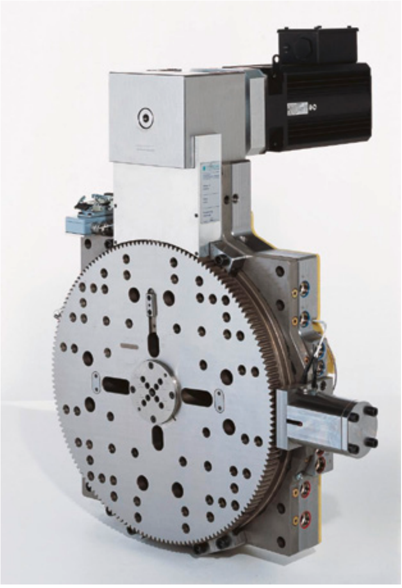

Figure 6.11 Rotary disk with servo drive (Source: FOBOHA)

Compared to the transfer technique, rotary molds have the advantage of better positioning of the preform in the finishing station. Even more complex design variations of the cavities are possible.

6.2.6 Paternoster

In the broadest sense, the paternoster works like the transfer process.
The parts are transferred into the next injection station through an integrated spindle drive.

The highlight of this mold technology is the use of standard multi-component injection molding machines without special specifications. The molds are slightly longer than swivel molds, but must not be rotated. This has the advantage that no plate enlargement or rail extension, as otherwise usual, is necessary.

Through a spindle drive, the preforms move into the next station. There, the parts are molded and transported to a mold removal station, which is outside the mold. A handling unit takes the finished parts during the following injection phase without a loss of cycle time. The empty mold halves are passed to the nozzle side. During the mold opening stage, a second spindle drive carries the empty mold halves from the removal to the pre-injection station. The cycle of the paternoster is closed, and a new process starts.

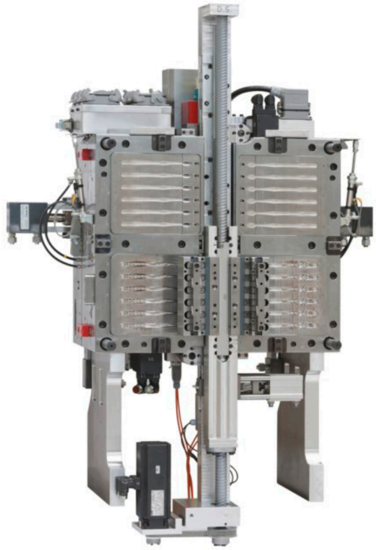

Figure 6.12 Paternoster mold (Source: Zahoransky Group)

Paternoster molds do not need to be rotated. They only need standard multi-component injection molding machines without special specifications. In each cycle, preforms and finished parts are injected at the same time. The finished parts are removed from the removal station, which is located outside the mold.

6.2.7 Cube Technology

The advantage of the cube technology over other mold technologies is that the number of cavities for the same machine size can be doubled. Or in other words: for the same production lot, the size of the machine is almost halved.

6.2.7.1 Stack Turning Technology

In the stack turning technology, the mold rotation is performed with a horizontal rotary device in the form of a center plate.

Preforms are produced in the first parting plane. When the mold is opened, they remain on the side of the moving center plate. For a fully opened mold, the center plate is rotated 180° and placed in the second parting plane. After closing again, the second component is injected in the second cavity where the preforms are located on the center block.

When using stack-turning molds with 4 × 90° rotation, stations 2 (operator side) and 4 (non-operator side) can be simultaneously used for further production steps. For example, station 2 can be used for cooling the molded parts and station 4 can be used for removal by a robotic system. Both functions take place without affecting the cycle time. Alternatively, station 2 can also be used for mounting (in-mold assembly) inside and outside the system.

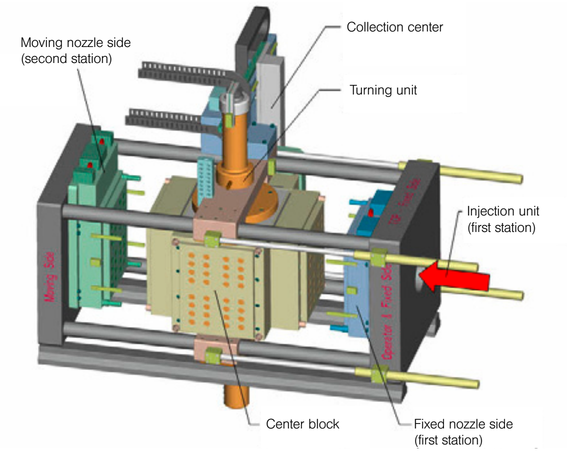

Figure 6.13 Principle of the stack turning technology (Source: FOBOHA)

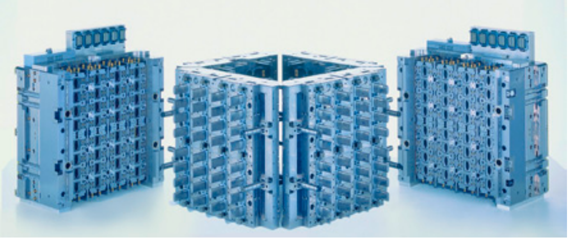

Figure 6.14 Stack turning mold (Source: FOBOHA)

When using the stack turning technology, the required clamping force is reduced by almost half. This mold technology is particularly useful for flat composite parts. In the 4 × 90° cube technology, the cycle time is reduced by approximately 25 % (compared to other methods) by simultaneous operations.

6.2.7.2 Double Cube

The double cube consists of a mold with two rotating cubes between its two mold halves. In principle, the double cube works like two separate molds. It has three clamping levels in which operations take place simultaneously. This allows production of complex parts with very high efficiency.

The double cube offers great advantages over conventional molds when it is necessary to relocate assembly operations into the mold and thereby saves a lot of cycle time. Assembly takes place simultaneously during the injection process in a closed mold.

The transfer of assembly operations into the mold is used more often. Additionally, the manufacture of parts with tight tolerances and therefore higher precision is possible.

Preferred applications for the double cube are the packaging, medical, and automotive industry. In a single operation, two or multi-part connections for packaging can be manufactured.

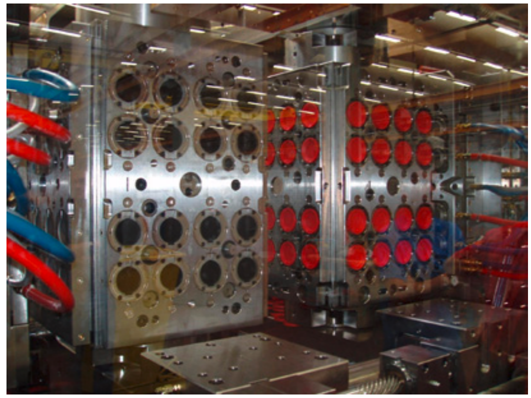

Figure 6.16 Double cube mold (Source: FOBOHA)

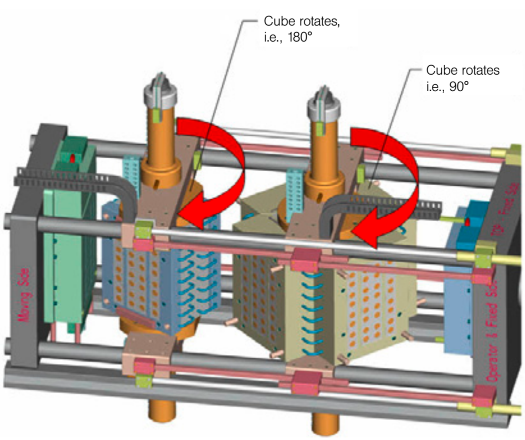

Figure 6.15 Principle of a double cube (Source: FOBOHA)

The costs of external mounting systems are eliminated by transferring the parts assembly into the mold. This reduces investment costs and space.

A high level of accuracy is ensured because parts of the same coordinated cavities are always interconnected.

The parts quality is significantly increased, because much tighter tolerances can be implemented compared with later parts assembly.

6.2.8 Tandem Mold

In the tandem molds, two molds are located behind the other. They are successively filled with plastic material and alternately opened in a cyclic manner.

In stack molds, the cavities are arranged one behind the other in several parting planes and are simultaneously filled with each shot; parts are simultaneously demolded after opening the mold.

With tandem molds, however, the parting plane opens cyclically staggered. This means that, while one mold half is cooling, demolding and re-injecting can be done in the other half. The dead time of the cooling is thus used for a second injection process. Since both mold halves run sequentially, different injection molded parts of one component family can be manufactured. Therefore, the injection molding machine must have a special program to determine the right amount of plastic provided for each parting plane.

Thick, identical parts with a long cooling time are also perfect for this technique.

An externally mounted locking system allows the cyclic alternating operation. The lock acts as a rack system. So it is possible, using an adapter plate, to convert two existing injection molds into a tandem mold.

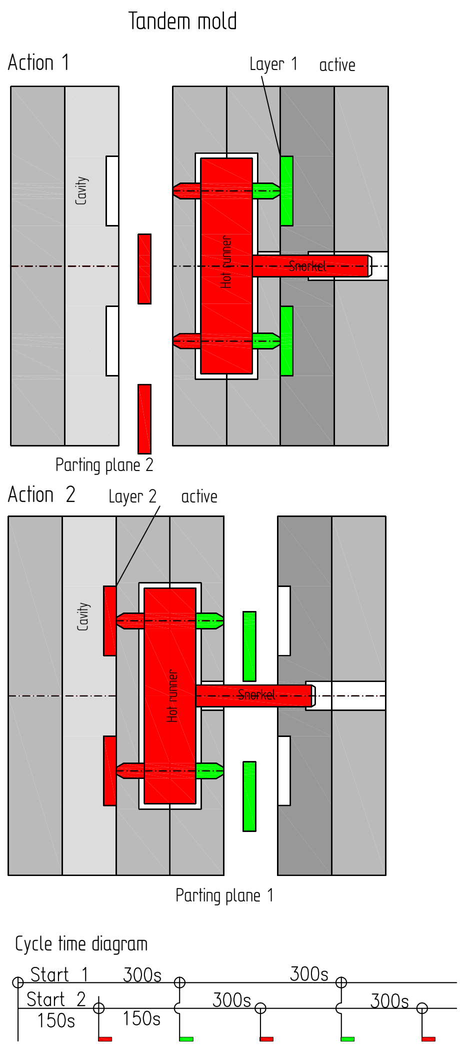

Figure 6.17 Principle of a tandem mold

With the tandem technology, the productivity can be doubled using a standard injection molding machine. Different injection parts of a component family are produced in cyclically displaced way.

6.2.9 Multi-Component Molds for Thermosets and Elastomers

In thermoset multi-component molds, hard-soft combinations are usually paired, although rarely are thermosets paired with thermosets. However, there are combinations with high temperature-resistant thermoplastics.

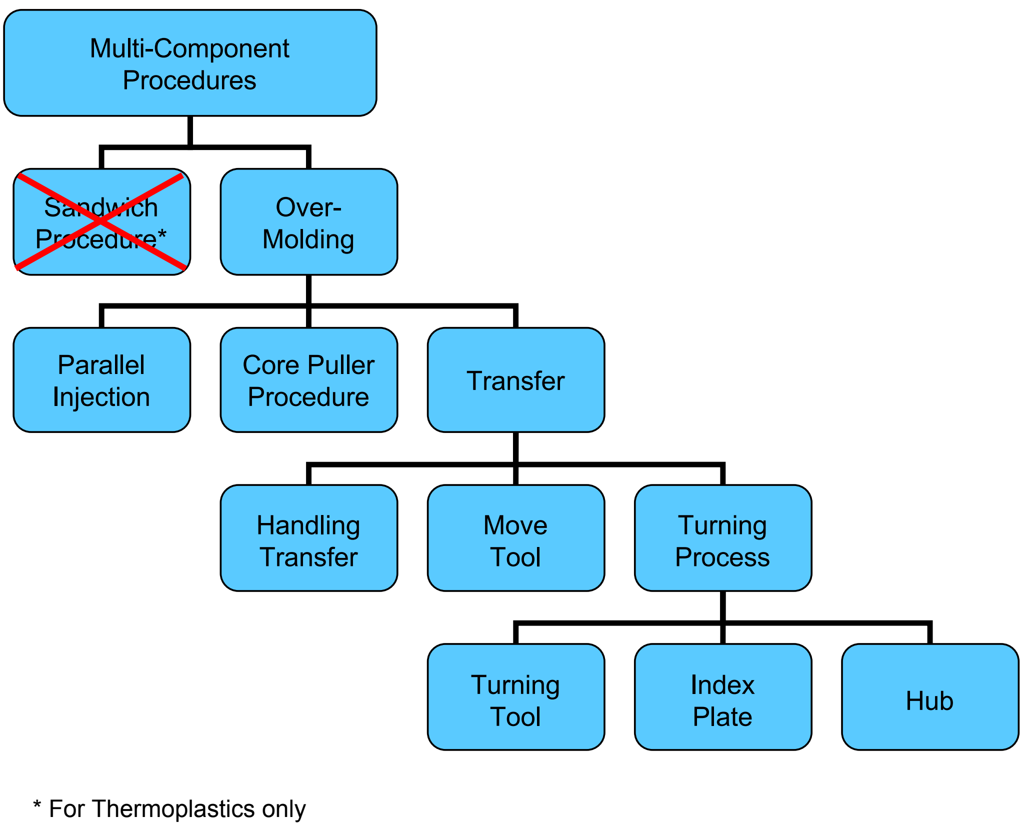

Table 6.2 Multi-Component Processes for Thermosets and Elastomers

Elastomers can be connected to both thermoplastics and thermosets. In both cases, the preforms have to be made out of hard components.

The combination of thermosets and elastomers is particularly suitable in the engine area. Both plastics are thermally stable and resistant to oils and fuels. Thermosets and elastomers (usually nitrile butadiene rubber [NBR]) can therefore be processed well with each other because these material groups are very similar. Both are processed with heated molds at approximately the same temperature level. The difference is that thermosets cure and elastomers vulcanize completely.

Hard-soft combinations are used to improve haptics or to absorb vibrations. Examples of improved haptic properties through hard-soft combinations are small machines, such as hand drills, welding guns, or hair dryers. In automotive engineering and in engine technology, such combinations can be used to selectively absorb vibrations.

Figure 6.18 Multi-component part (Source: Ferromatik Milacron)

The most common multi-component applications are hard-soft combinations with elastomers. The application areas are housings with excellent haptics or technical parts to reduce vibrations in automotive engineering.

6.3 Thin Wall Molds

Thin wall molds are high-speed molds that are primarily used in the packaging sector. The molded parts have typical wall thickness of less than 1.0 mm.

Thin wall technology sets particular requirements on the mold. Due to the rapid cycles and the high volume of parts produced, the steel selection, the core centering and the venting during injection are of extreme importance.

Thus, for example, so-called floating cores with a wide core base are used for multi-cavity molds. This compensates for the expansion of the mold cores by the resulting heat. The mold must be manufactured from pre-hardened steel. The mold inserts (mold cores and dies) are through-hardened.

Critical to the quality of the molded parts with short cycles is the venting of the mold during the injection phase. Depending on the part, the air from the high injection speed must be displaced from the mold cavity within 0.1 seconds. Therefore it is necessary to incorporate venting ducts into the parting plane around the mold cavity. Experience has shown that 0.5 to 1.0/100 mm, and 3 – 4 mm wide channels are sufficient to dissipate the air. If air removal is not sufficiently taken into account, there is a danger that the plastic mass will burn.

Thin-walled parts are preferably found in the packaging industry, in medical technology, and in telecommunications. Typical applications include containers with a wall thickness of less than 1.0 mm, such as yogurt cups, margarine boxes, ice containers, mobile phone cases, and plant pots with very thin wall thickness of approximately 0.4 mm.

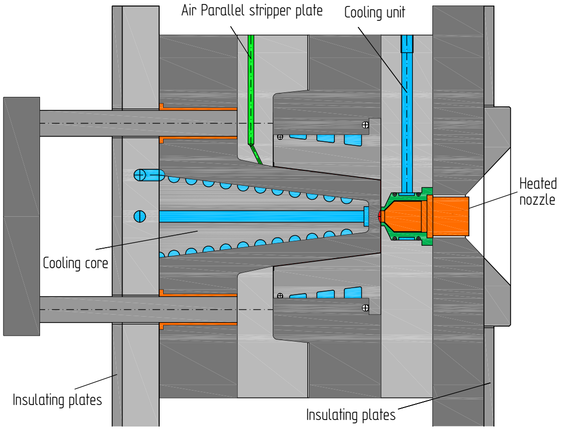

Figure 6.19 Principle of the thin wall mold

Packaging parts with wall thicknesses less than 1.0 mm with a flow path/wall thickness ratio of 150 : 1 can be manufactured with thin wall molds. However, thin-walled molded parts with a flow path/wall thickness ratio of 450 : 1 also belong to this special design.

6.4 Insertion Technology

In the insertion technology, parts made of materials other than plastic, often metal, are supplied to the mold and overmolded with plastic.

In most cases, these insertion molds run on vertical injection molding machines. They are very often equipped with a sliding or rotary table. The basic variant of the mold mostly consists of an upper mold part (the vertical machine) and several lower mold halves. Thus, it is possible to simultaneously remove the finished workpieces from the closed machine and equip the now free lower mold part with new inserts without loss of cycle time.

The most complex aspect of the insert technology is the automatic feeding and fixing of “nonplastics” in the mold, preferably metals. With appropriate handling devices, this process takes place almost completely automatically.

The inserts always have undercuts, notches, and grooves on the knurls, which guarantee a permanent bond with the plastic.

The most common applications are in the electrical and medical technologies: threaded bushings, plug contacts, cable strands, screwdriver handles, and hypodermic needles are typical inserted parts.

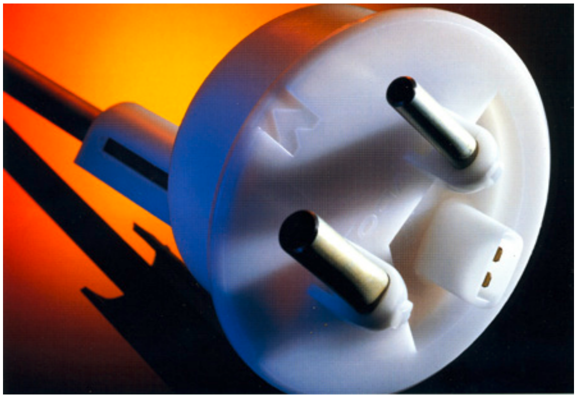

Figure 6.20 Sample part insertion technology (Source: Ferromatik Milacron)

The insertion technology has its advantages in complicated designed inserts. Precise metal threads in the plastic part, stiffeners (if required), or enclosures of metal parts with plastic can be produced perfectly with this method.

6.5 Fluid Injection Technology

6.5.1 Gas Injection (GIT)

In the GIT technology, for thick-walled parts, the plastic core is blown free with gas. As a result, the injection molded part becomes lighter and stronger, and the cooling time is greatly reduced.

Generally, the injection process consists of three stages: injection, holding pressure, and cooling. In the GIT-technology, the function holding pressure is taken over by the gas pressure. The process is similar to inflating a balloon. The molded parts are hollow on the inside, and thus thin-walled.

In the GIT process, attention should be paid to some mold-specific peculiarities. The gating system has to be a cold runner because gas is led through the plastic core of the runner system to the mold part. Generously sized gate cross sections are advantageous. The gas used is nitrogen, because oxygen would burn the plastic under pressure.

Different methods of introducing the gases are offered by the machine manufacturers. This can be done via the machine nozzle or several separate mold nozzles.

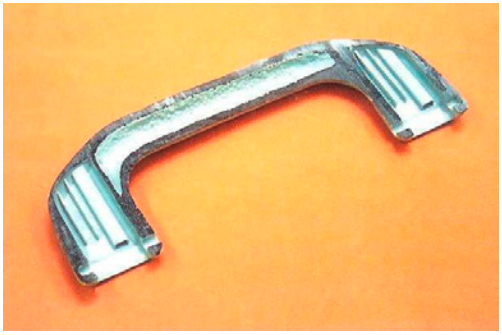

Figure 6.21 Molded part from fluid injection technology (Source: Ferromatik Milacron)

Advantages of fluid injection are a high degree of design freedom of the parts, short cycle times with thick-walled molded parts, better mechanical rigidity, uniform shrinkage, less residual stress and warpage, reduction of sink marks, and lower demolding forces.

6.5.1.1 Partial Filling

In this method, the molded part is filled until the gas completely fills the molded part with the material to be displaced. After completely filling the molded part, the gas is introduced via the machine nozzle. The gas holding pressure remains in effect until the molded part is dimensionally stable. After solidification of the melt, the gas pressure is built up. The gas is allowed to escape into the environment. Newer approaches include a gas recovery system.

Because the gas is introduced via the machine, every injection mold with distribution channels can be used for the GIT technology.

With the gas injection technology, thick-walled parts such as gas pedals, door handles and grips for cars, wiper arms, and clothes hangers are manufactured.

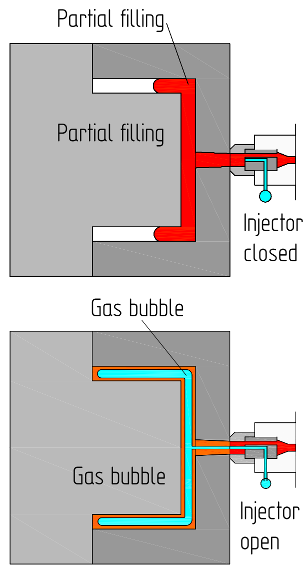

Figure 6.22 Principle of partial filling

6.5.1.2 Secondary Cavity

In this method, the molded part is also 100 % filled. The material to be blown out is passed into a so-called secondary cavity, which is open at the start of blowing. The molds used are special molds. Hot runner molds cannot be used.

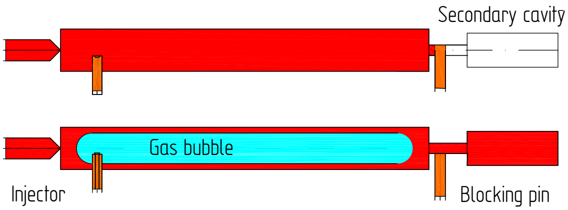

Figure 6.23 Principle of a secondary cavity

Standard molds with distribution channels can be used. Since these do not have a gas injector in the mold, more expensive special nozzles with gas injectors are needed. Changeover marks form at a standstill of the mass.

6.5.1.3 Blow-Back Process

In the blow-back process, a special mold is required. The molded part is completely filled. Then blowing out takes place. It is crucial that the blown out material is led back into the injection unit.

The amount of blown plastic is controlled through the path in which the melt runs back. The blow-back process has advantages, as no material is lost and no changeover marks form, although the process is not always usable.

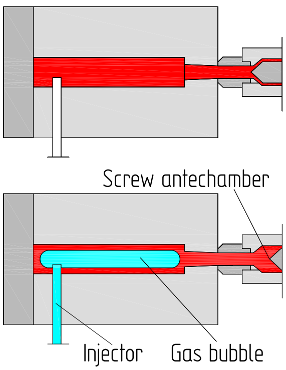

Figure 6.24 Blow-back process

6.5.1.4 The Core Pull-Back Process

This technique is primarily used when thick-walled sections are to be blown partially.

Typical applications include handles and laundry baskets, which have very large wall thickness differences and a tendancy to warp and distort.

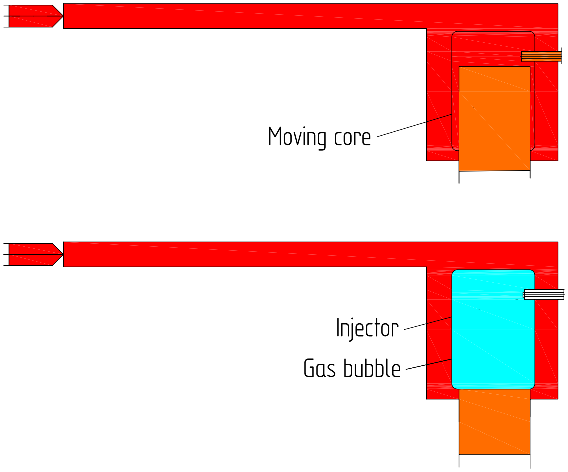

Figure 6.25 Core pull-back process

6.5.2 Water Injection (WIT)

WIT-technology has advantages over the gas injection technique in that it reduces cycle time by shortening the cooling time, reduces the material used in the manufacture of polymeric hollow bodies, and does not require gas.

In the WIT-technology, water displaces the melt in the form of a piston. The spontaneous cooling at the parting plane between the melt and the water leads to a film formation of solidified melt. The water pushes the still liquid melt before it, which creates a small wall thickness. The excess melt is pressed back in the screw antechamber through a secondary cavity. Blowing off of the water is done, depending on the injector type, outside of the component or by an automatic water recirculation within the injection cycle.

The injector is the most important connection between the WIT unit and the injection mold. Depending on the application, different types of water injections can be used to initiate the water at any point in the molded part.

In the area of strong curvatures, strong differences in wall thickness result in the WIT method; extreme deflections should therefore be avoided.

A number of parts are manufactured with the WIT method, preferably those with hollow bodies; they include handles, pedals for vehicles, household appliances, office furniture, and sports equipment.

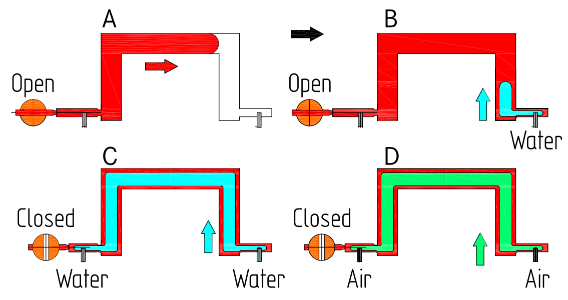

Figure 6.26 Principle of WIT (Water injection technology)
Phase A: Fill the cavity
Phase B: Introducing water
Phase C: Rinsing with water
Phase D: Blowing out with air

Significant advantages of the water injection technology in comparison to the gas injection technology are the shorter cycle times and lower residual wall thickness of hollow bodies, as well as a reduction of the component stress.

6.6 Push-Pull Injection Molds

In push-pull injection molding, two material flows of the same plastic material are pushed through the injection molded part in opposite directions.

If a molded part has to be constructed with two or more injection points, the component is weakened by the weld lines. Weld lines occur whenever two material flows in the mold meet. The two edges penetrate but create structural damage.

The development of weld lines is prevented during push-pull injection molding due to the opposing flow through of the plastic material. The molds are designed so that the mold cavities have two injection points, ideally on opposite sides.

Injection unit 1 fills the mold cavity through gate 1 and forces the material back into injection unit 2 through the second gate.

Now the push-pull process begins. Unit 2 injects against unit 1 and pushes material through the molded part back into unit 1.

This process can be repeated until a completely weld-line-free molded part is formed.

The push-pull injection process also brings a significant improvement in quality in the processing of glass-fiber-reinforced materials and newer materials such as liquid-crystal polymer (LCP) and polyether ether ketone (PEEK).

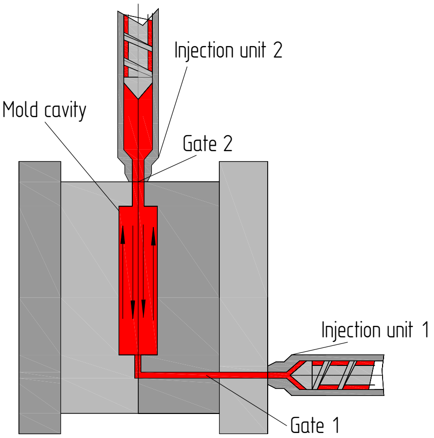

Figure 6.27 Principle of push-pull injection molding

Push-pull injection molding is used to prevent weld lines and thus breakage of the molded parts under load.

6.7 Implantation Injection Molding

Implantation injection molding is a technology that is similar to cascade injection molding, but the injecting is done with two components.

Implantation injection molding is used when partially on or in a large-scale injection molded part, one piece of a different material is injection molded. It is only suitable when the second component must not be clearly defined. The two material flows meet depending on how delayed the second mass was injected. Through the delay time during the injection of the second component, the position and the size of the spots can be determined. Therefore, the boundaries between the two components are blurry.

The molds are relatively simple. The only difference from a conventional cascade mold is the second hot runner for the second plastics material. Although the separation of the two components is created arbitrarily, it is nevertheless a two-component injection molding process. A rotary table is not needed for implantation injection molding.

Typical applications include engine covers, which require a vibration-damping, elastic support.

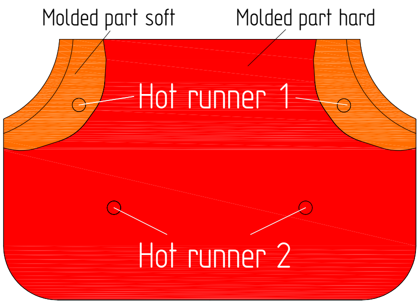

Figure 6.28 Principle of implantation

Implantation injection molding is the cheapest way to manufacture two-component injection molded parts. It does not require separate cavities for the preforms nor a rotating device and therefore requires no rail extension.

6.8 In Mold Labeling (IML) Process

Thin-walled plastic packing material and plastic cards are the preferred applications of the IML process. By inserting a printed film directly into the mold, a completely printed injection molded part is created in one production step.

In the IML process, pre-punched decor labels are preferably inserted into the mold and are back-molded. The supply of the labels from the magazine to the mold cavities is done by a handling device. The labels are picked up by the gripper of the handling device by vacuum. Holding the labels in place in the mold is done by means of a vacuum or by electrostatic charge.

In the design of the mold, a suitable needle shut-off system must be taken into consideration, because high temperature through friction at the nozzle tip can cause damage of the labels.

A corresponding stiffness of the film is necessary for a good grip of the labels. The thickness should be about 50 – 80 microns.

Figure 6.29 IML injection molded part (Source: H. Müller Fabrique de Moules S. A.)

The IML process makes subsequent printing or application of adhesive labels superfluous. The back-molded label provides additional stiffness of the molded part, which often allows a reduction of the wall thickness of a thin-walled packaging element. A label changeover is easy and can be done without interrupting the production.

6.9 Cascade Injection Molding Process

In cascade injection molding, the gate points of the hot runner molds can be filled one after another, step-by-step.

This technology is used to overcome long flow paths or when viscous plastics have to be processed.

The flow paths are divided into several sections. Each segment of this division is molded with a hot runner nozzle, which is designed as a needle shut-off nozzle. In cascade injection molding, not all hot runners inject simultaneously but are sequentially switched on or off. The molded part is filled step-by-step.

The material stream flows continuously. If the injection molded part is filled, holding pressure builds up, which then acts uniformly over all nozzles.

The internal mold pressure results from the flow path as a function of the wall thickness and the melt flow index (MFI). From this pressure, multiplied by the area of the molded part, the necessary closing force can be calculated (see Section 1.4 on buoyancy forces). It follows that the machine always clamps using only the force of the part that is being injected.

The cascade injection molding is used in the manufacture of components for the automotive industry. Typical parts are bumpers and trims. Cascade injection molding is not suitable for producing optical parts because the changeover marks are visible.

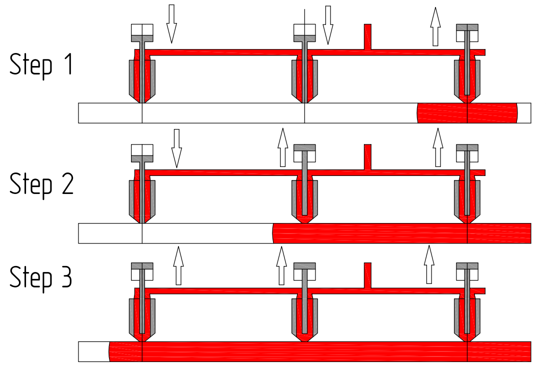

Figure 6.30 Principle of cascade injection molding

Cascade injection molding can only be done with hot runner molds in connection with needle shut-off nozzles. Only when the melt front reaches the adjacent nozzle, is the mold opened. Simultaneously, the nozzle in front of it closes and weld lines can be avoided.

6.10 Lost Core Technology

Hollow parts with complex geometries can be produced with the lost core method, in which the cores cannot be pulled over the slider anymore.

In this method, a core, which is the interior of an injection molded part, is manufactured from a bismuth alloy in the pressure casting process. This core is inserted into a mold and injected with a plastic material.

The core, which is now enclosed in plastic material and located inside of the molded part, must now be removed. Therefore, the molded parts are immersed in 80°C hot water. The core melts because the bismuth alloy only has a melting point of 70°C.

The melted alloy collects in the form of beads at the bottom of the water tank. If the beads are dried, they are reused for new cores. Since bismuth gives a clean and smooth surface during casting, the molded parts look like they are polished inside.

As an alternative to the lost core method, when possible, shells are injected and subsequently glued or welded. This is a much cheaper option.

One of the main applications of this technology are intake manifolds for the automotive industry, cooling and fuel-carrying parts of cars, as well as pump housings, paddle wheels, and plumbing parts.

Figure 6.31 Intake manifold (Source: Ferromatik Milacron)

Using this method, complex parts with nearly closed cavities can be produced. If technically possible, the dual-shell technology is preferred today because it is less expensive.

6.11 Material-Dependent Special Processes

Material-dependent special processes can be done on existing molds. Little or no modifications have to be made at the mold itself.

6.11.1 Marbling

Today, there are only a few applications of marbling. All of the molds with a cold runner system can be used without restriction.

Figure 6.32 Soap holder with marbling effect (Source: FERROMATIK MILACRON)

6.11.2 Micro-Foam Injection Molding (MuCell)

In this method, propellant gas is introduced into the front part of the cylinder. A special mixing zone causes microcells to form in the plastic. All types of molds can be used without an additional device. Hot runner molds must be equipped with needle shut-off nozzles.

6.11.3 Thermoplastic-Foam Casting Process

In this method, propellant is added to the plastic granules, frequently in the form of granulate, or sometimes in liquid form.

This application is appealing for parts with wall thicknesses from 20 mm. All common mold designs and gate systems are usable. Hot runner systems require a shut-off needle. Since no high injection and holding pressures are necessary, molds made from aluminum are in some cases used.

Figure 6.33 PVC-Fitting (Source: FERROMATIK MILACRON)

6.11.4 PVC Processing

In the processing of polyvinyl chloride (PVC), hydrochloric acid is released. For this reason, the molds have to be protected against corrosion.

All types of gates except hot runner systems are used. Since PVC is sensitive to shearing, the runners have to have rounded deflections. Because of the high injection pressures, only hardened molds should be used. The gate dimensioning should be designed more generously than in other thermoplastics to avoid friction. The molds must be uniformly heated to about 40 – 50 °C.

6.11.5 Monosandwich Process

In this method, extruding into the injection cylinder is done with a laterally arranged extruder via a feeder material. In the screw antechamber, the two material blocks, the skin material from the secondary extruder and the middle layer (often a recycled material), are arranged in a row. All types of molds, except hot runner molds, can be used.

Figure 6.34 Car handle in the mono-sandwich method (Source: FERROMATIK MILACRON)

6.11.6 In-Mold Painting

In-mold painting means that the injection molded parts leave the mold with a high gloss as though coated with a lacquer. This method is used for plastic parts that should have a decorative and scratch resistant surface.

Molding and ejecting is done on the back side to ensure that no markings are visible on the face side. All gate variants can be used.

Typical applications include bumpers, fenders, cell phones, and cosmetics packaging.

6.11.7 In-Mold Welding

In this method, plastic shells are welded together by overmolding. Therefore, the part halves are either injected in the mold, and then connected by injection molding, or the parts are inserted into the mold and then tightly welded together through injection molding.

6.12 Micro-Injection Molds

According to the the Institute of Design and Production in Precision Engineering in Stuttgart, Germany, a distinction has to be made between micro-injection molded parts and microstructure components.

Microstructure Components

Microstructure components are injection molded parts with standard dimensions, but which have one- or two-sided zones with microstructure and a mass in the range of several grams. A well-known application example is CD and DVD storage media.

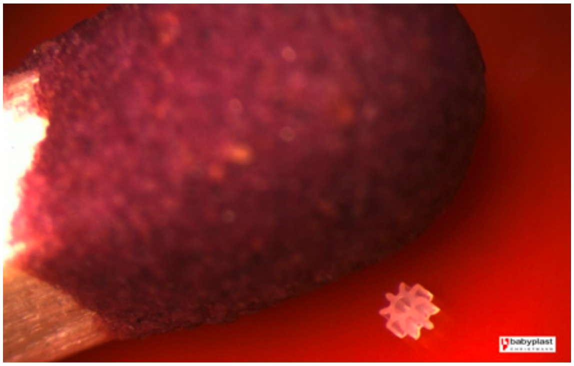

Figure 6.35 Microstructure part (Source: Christmann, Kunststofftechnik)

Micro-Injection Molded Parts

Micro-injection molded parts have shot weights in the milligram range and dimensions of a few millimeters. Small cavities like this cannot be produced with conventional surface treatments, such as milling and electrical discharge machining (EDM). The mold cavities are formed by electroplating or by etching. The X-ray lithographic electroplating (LIGA) process is particularly suitable.

Temperatures of the plastic material and mold have a decisive influence on the quality of the micro-injection molded parts. The mold should be heated by an additional heater. After the injection, the cavity is cooled to demolding temperature. This process is called variothermal process control. Another important criterion for the manufacture of high quality parts is an optimal evacuation of the air from the cavities during the injection process. If this is not guaranteed, the mass burns and dark spots form.

Micro-injection molded parts are used particularly in the watch industry, communication technology, medical and biotechnology, and in the area of sensor technology.

Micro-injection molded parts are parts in the milligram range with dimensions of a few millimeters. The molds are equipped with a special heating/cooling system, called variothermal process control.

6.13 Powder Metal/Ceramic Molds

Powder injection molding uses the technology of injection molding for molding complex components made from metallic or ceramic materials.

For the manufacture of the materials, the ceramic or metallic powder is mixed with a binder (binding, wetting, and lubricating agents). This plastic mass, which is available in granular form, is processed on conventional molds (in so-called green bodies) in injection molding. Due to the high amount of abrasive materials (approximately 50 – 70 %), the molds should be through-hardened.

The gating systems are the cold runner systems, which separate in the mold. The molds have to be heated because of the good thermal conductivity of powder metal parts. Since the material is highly sensitive to shearing, there is a danger that it may decompose by local overheating of the binder. This would mean that the powder materials are not distributed evenly in the mold, which leads to errors in the finished part.

Although the injected green bodies are dimensionally stable parts, they are still very fragile. During removal, larger parts can fall out of the mold from their own weight and break.

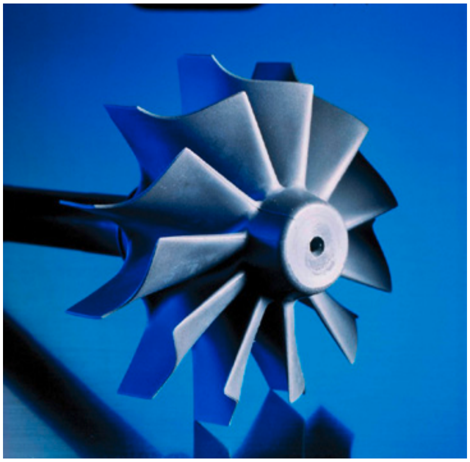

Figure 6.36 Metal Injection Molding (MIM) sample part (Source: Ferromatik Milacron)

The binder is then removed from the green body. This so-called "debinding" is carried out in a sintering furnace. Here, the green body is heated above the decomposition temperature. The decomposition products escape as a gas. The result is a porous powder structure that has sufficient inherent stability that its geometry remains accurate. The green body has now become a brown body.

Usually, a small amount of binder remains in the part. In the further course of the process, the parts are placed into the sintering furnace. Depending on the powder material of the part, the sintering temperature can be up to 2000 °C. During sintering, the molded part is compressed to up to 98 %. In this process, the parts lose 15 – 25 % volume fraction, depending on the binder content.

Depending on the binder mixture, debinding takes a few hours up to several days. The time is dependent on the wall thickness of the components. A wall thickness of 10 – 15 mm should not be exceeded for cost-effective debinding.

The advantage of powder injection molding is the high production quality and the accuracy of the molded parts. Furthermore, the material to be processed can be precisely tailored to the product. The only disadvantage is that the injection-molded part cannot be produced in a single-step process.

Typical applications include ball bearing rings, gears, locking cylinders, cutting elements, and parts for the medical technology area.

Figure 6.37 Ceramic Injection Molding (CIM) sample part (Source: Arburg)

6.14 Rapid Prototyping

This term first appeared in the early 1990s. It means either the production of dimensionally accurate and functional plastic components or the production of mold inserts.

Rapid prototyping means rapid manufacturing of sample parts instead of first building a prototype mold. Sample parts are directly produced through stereolithography data. For this, it is necessary to convert the present 3D data parts drawing into stereolithography data (STL). With this data, a dimensionally accurate and functional plastic part can be created.

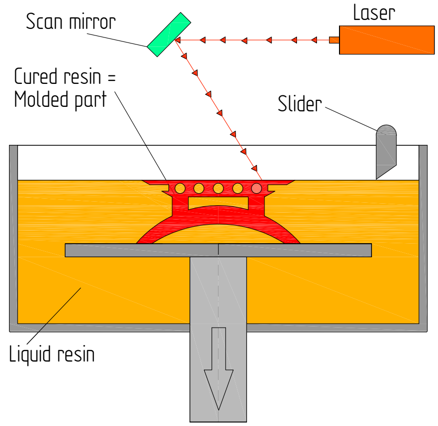

Figure 6.38 Principle of the stereolithography

Sintering Process

Further development in the field of sintering processes and sintering materials opened up new ways for producing molded parts. With these sintering facilities, components made from plastics, such as polyamide, acrylonitrile butadiene styrene (ABS), and polyester, but also from metal, could be manufactured in the melting or sintering process. Different materials are available in the metals sector. These include, for example, steel, stainless steel, mold steels, titanium, and aluminum. These raw materials must be available in powder form.

The properties of components made from sintered metals are almost comparable with mechanically manufactured parts. For the sintering process, the 3D computer-aided design (CAD) data is converted into a STL data set.

Selective Laser Sintering (SLS)

Selective Laser sintering is a generative layer construction process by which any three-dimensional geometry can be produced. In SLS, a metal powder is locally melted and interconnected though a laser or an electron beam. The sintering or electron beam is controlled by the STL data.

With the sintering process, molded parts are manufactured in a simple way. With these molded part inserts, prototype sample parts can be produced quickly and cheaply. This method is not suitable for series production.

A laser sintering system consists of a laser, a scanner, and a container that holds a photo polymer or epoxy resin. With a controlled laser beam, which develops heat upon touching the polymer layer, liquid polymer components become solid materials.

The laser beam draws, as with a pen, the contours of the component to be produced into the polymer. When the first layer is laser cut, the container is lowered by a few tenths of a millimeter, and this process is repeated until the component is completed.

Selective Laser Melting

In this method, metal powder is distributed on the working plate in a layer thickness of 0.15 mm. At the points where the laser beam strikes, the powder particles are bound to each other by melting and form a layer of the final component. The work table is lowered and another powder layer is applied. The sintering process starts again. The finished component is created layer upon layer. The finished component is integrated into a prototype base mold. Small series of molded parts can be produced with this.

The properties of components made from sintered metals are almost equivalent to those of mechanically produced parts.

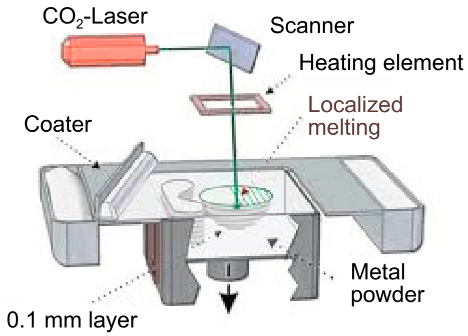

Figure 6.39 Principle of rapid prototooling (Source: The University of Applied Sciences Munich, faculty 03, Florian Schneider)

Parts manufactured using rapid prototooling can be used as a substitute for pressed or forged parts and have comparable mechanical properties.

6.15 Rotary Table Molds

When an injection molding cycle takes a great deal of time because the cooling time is extremely long or parts have to be inserted, rotary table systems are a good alternative.

If parts have to be inserted, an upper mold half and multiple lower mold halves are used. The ejectors are incorporated into the lower mold halves. An injection station, a feed station, and an ejection station are available. The injection unit is arranged horizontally and is aligned with the parting plane of the molds. The material is injected into the parting plane.

Inserting is done into the lower mold half, which is free-standing and on the opposite side of the injection unit.

The rotary table technology has the advantage that inserting can be done into the free-standing lower mold half, preventing the parts from falling out of the mold. To take advantage of this system in a meaningful way, four or more stations are used.

In injection molded parts with extremely long cooling times, another rotary table system is used. There are several molds, consisting of upper and lower half on the rotary table, which slowly cool from cycle to cycle. The injection molding machine is equipped with an injection and ejection station. Injection is done in the parting plane.

The ejection station is also at the same time the removal station. The other molds remain closed. The more molds that are used, the shorter the cycle time will be.

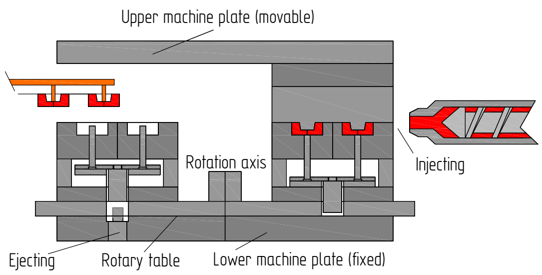

Figure 6.40 Principle of a rotary table mold

Rotary table molds have the advantage of short cycle times; a disadvantage is the high tooling costs.

6.16 Silicone Molds

Silicone molds are electrically heated; they have a vacuum connection, and the runner is designed as a cold runner.

After a close look at the molds, one finds that the gates are thinner than those of thermoplastic or thermoset molds. The molds must be worked with an extremely high precision, because the low-viscosity, liquid mass flows into gap widths that can be 0.01 mm and smaller. Venting through the parting plane is therefore very difficult.

These molds should not be polished. Silicon parts tend to stick in the mold cavity. This is especially true for new molds. This phenomenon disappears when the molds are in operation for one to two hours.

In the selection of materials, which are combined with liquid silicone rubber (LSR), it should be noted that they will remain dimensionally stable at the mold temperature of up to 180 °C. Materials from the group of the polyamides with a proportion of up to 50 % glass fibers are mostly used.

All mold types known from two-component thermoplastic processing can be used.

Figure 6.41 Bottle nipple (Source: FOBOHA)

Many LSR (silicone elastomer) parts cannot be used without stabilization in a strong plastic frame made from thermoplastic. This two-component technology is very expensive and therefore only suitable for large quantities.

6.17 Injection Blow Molds

Injection blow molding consists of two manufacturing steps. In the first step, a preform is molded in the injection molding process. In the second step, which takes place subsequently or simultaneously, the preforms are inflated in a blow mold.

To transport the preform from the injection station into the blowing station, a transfer movement is needed. Therefore, a central block, which can be pivoted and moved, is integrated. The mold concept is relatively simple. There are dies for the preform and at another station dies for blowing. The cores are the same for both stations.

To achieve short cycles, index plate molds are used depending on the injection blow molded part. These are equipped with a third station used for cooling and for stripping the parts. The injection blow molded parts achieve the same quality as pure injection molded parts.

Common areas of application of injection blow molds are:

- The packaging and pharmaceutical industries.
- Wide-mouth containers, cans, bottles, and other containers.

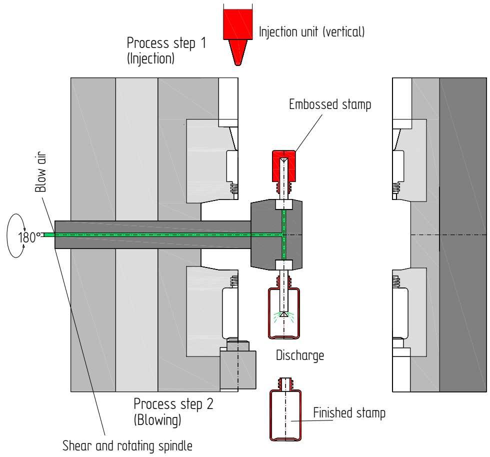

Figure 6.42 Principle of injection blow molding

Figure 6.43 Blow molded parts (Source: Uniloy Milacron)

Injection blow molds are equipped with a center block in order to transport the preforms into the blow station. Depending on the part, they can be stripped in another station for demolding of cores.

6.18 Injection Compression Molds

Injection compression molding is used for the manufacture of optical components. The most popular applications are the manufacture of magnifying glasses, lenses, lens blanks, and optical data media.

In this method, the plastic melt is injected into the mold, which is only opened a few tenths of a millimeter to a few millimeters. In the subsequent compression phase, the mold closes and compresses the plastic in the mold with the set clamping force of the machine. Thereby, a uniform pressure is exerted on the entire cavity blocks. The low pressure, which is required for filling, prevents voids and sink marks and reduces internal stresses and warping effects, and double refraction can be minimized. The molded parts are almost free of residual stress.

CD blanks are produced under this process. The digital information, which is located on the female mold half is correctly transferred to the CD blank by injection compression molding.

Depending on the part geometry, injection compression molding can be limited to part sections. Regarding the mold, it should be noted that the gate remains closed during the compression process, so that the molten plastic material in the mold cavity does not push back into the screw anteroom. The mold should also have shut-off edges to define the mold cavity.

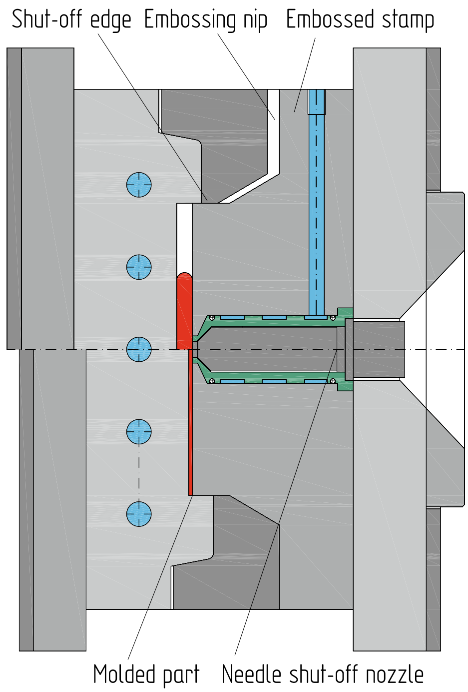

Figure 6.44 Principle of a compression mold

Internal stresses and double refraction in optical components can be avoided with the injection-compression molding process. During compression, a low pressure is exerted on the entire cavity surface.

6.18.1 Injection Compression Molds in Elastomers

In simple molded rubber parts, which are manufactured without special requirements in large numbers, injection-compression molding processes are used.

In this method, the elastomer mass is pressed into the mold, which is already slightly opened or which opens during the injection. This is done with a low injection pressure. Here, the mass outgases very well. The machine closes when the dosed mass is injected, and the mass is pressed through the distribution channels and through the sprue into the mold cavities. In elastomeric injection compression molding, material is distributed over the entire mold surface in the area of the molded parts. This is called the sprue.

The vulcanized parts are then ejected, attached to the sprue, or removed with an extraction device.

Rolling and sealing rings in the wastewater sectors or vibration dampers, membranes, or similar rubber parts are typical applications.

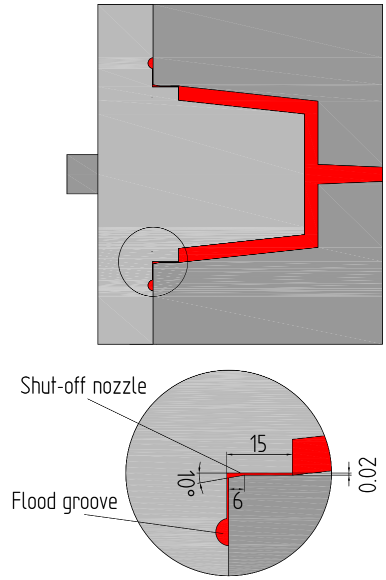

Figure 6.45 Principle of injection compression molding

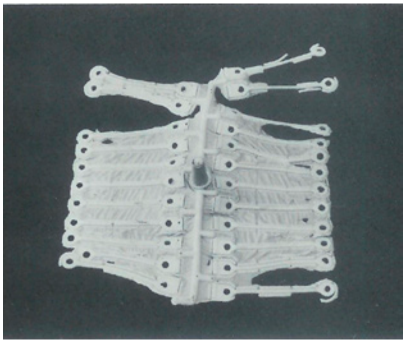

Figure 6.46 Molded part with sprue

The molded parts hang in the sprue. They are either punched from the sprue or shock frozen and then tumbled. The deep freezing temperatures during shock freezing breaks the thin sprue skin and the parts are free of burrs.

6.19 Textile Back Injection Technology

In the back injection technology, decorative material, mostly textile or foil, is combined with plastic parts in the mold. Decorated molded parts without using adhesives are manufactured in one single step.

The adhesion of the decorative material takes place by the penetration of thermoplastic melt into the generally three-layered decorative material. To prevent damage of the decorative material during the ejection, the ejector pins are integrated on the nozzle side of the mold.

Critical to the textile back injecting is an ideal decor supply. The fabric is inserted, positioned, and clamped with clamping frames, needle grippers, and various brackets. By means of a handling device, the prefabricated material blanks are introduced into the parting plane of the mold. Then back injecting takes place.

In elongated components (e.g., car pillar trims), cascade injection molding is used. The injection mold must therefore be equipped with a hot runner system with needle shut-off nozzles. In cascade injection molding, the nozzles are sequentially opened or closed.

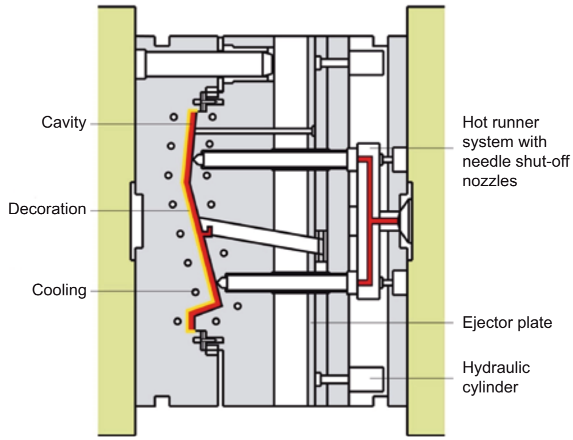

Figure 6.47 Principle of back injecting (Source: Georg Kaufmann Formenbau)

Figure 6.48 Molded part: textile is back injected (Source: Georg Kaufmann Formenbau)

In textile back injecting, textiles are firmly bonded with plastic parts in one process step. The time-consuming and polluting adhesive lamination of plastics is omitted.

6.20 Workpiece Carrier System

Workpiece carrier systems are mold halves or removable disks, which are clocked, lined up on a chain or a turntable through the injection molding machines.

The workpiece carrier system offers the advantage that the injection molding machine can run a fully automatic cycle. At the same time, supply parts can be inserted or the finished parts can be removed from the outside standing removable disks, which run in an infinite loop.

These systems can be used on both vertically and horizontally closing machines. The nozzle-sided mold half and the ejector is permanently installed in the machine.

Small parts are very often manufactured on such carrier systems; parts include electronic components (e.g., microswitches, LEDs, transistors) or parts for the medical sector.

A typical process flow for medical parts is:

- Insert needles for disposable syringes outside the mold.
- Mold the stroke.
- Insert the needle protection cover.
- Remove the finished parts.

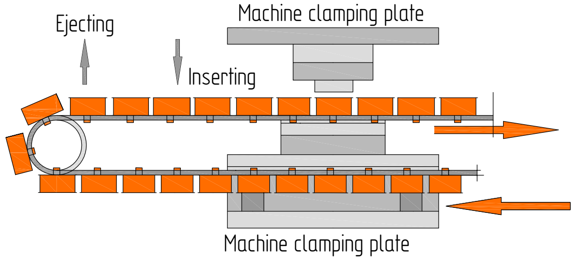

Figure 6.49 Principle of a mold carrier

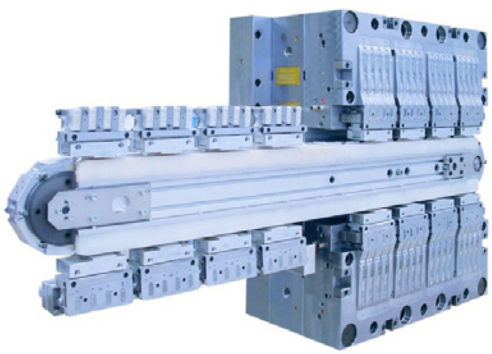

Figure 6.50 Mold carrier (Source: Zahoransky Group)

Advantages of the workpiece carrier system are a low cycle time, independent inserting and removing possibilities, and a high degree of freedom when inserting components.

References

1. N.N., “Spritzgieß-Schnellläufer mit Tandemwerkzeugen,” *Kunststoffe* March (2003) Hanser Publishers, Munich
2. N.N., “Mehr Farbe und Funktion,” *Kunststoffe* June (2003) Hanser Publishers, Munich
3. Johannaber, F., Michaeli, W., *Handbuch Spritzgießen* (2004) Hanser Publishers, Munich

7 Mold Surface Treatment

7.1 Common Surface Treatment Processes

All methods of surface treatment in mold making have the aim to reduce either the wear (abrasion) or the corrosion of the mold. All moving mold components are affected by wear. Corrosion prevention is about protecting the surface and the parts that come in contact with the cooling media. Wear and corrosion are mostly a gradual process but one with major economic consequences.

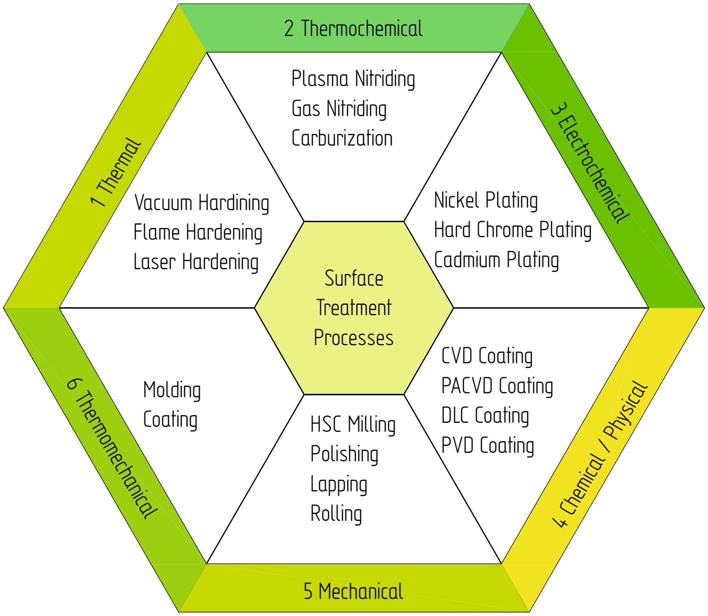

Figure 7.1 Typical Surface Treatment Processes

Table 7.1 Processing Guidelines of Common Surface Treatment Processes (Source: Hanser)

Process	Hardness range			Common process temperature in °C	Common layer thickness (µm)	Reproducibility	Dimensional accuracy	Preferred steel concept
	1000	2000	3000					
Flame hardening				*	up to 5000	+	+	Tempered steel
Laser hardening				*	up to 2000	+ +	+ +	Tempered steel
Case hardening				920°C	up to 2000	+ +	+ +	Case hardening steel
Chemical nickel				80	up to 20	+ +	+ +	Pre- and through hardened steels
Hard chrome plating				60	8 - 50	+ +	+	Pre- and through hardened steels
Gas nitriding				500 - 550	up to 50	+ +	+ +	Pre- and through hardened steels**
Plasma nitriding				300 - 550	up to 30	+ +	+ +	Pre- and through hardened steels**
Boriding				850 - 1000	up to 60	+ +	+	Pre- and through hardened steels
CVD/TiC				950	up to 10	+ +	+	Through hardened steels**
PACVD/DLC				200	up to 3	+ +	+ +	Through hardened steels**
PVD/CrN				220 - 450	up to 10	+ +	+ +	Pre- and through hardened steels**
PVD/TiN				220 - 450	3	+ +	+ +	Through hardened steels**
PVD/TiCN				450	3	+ +	+ +	Through hardened steels**
PVD/TiAIN				450	up to 10	+ +	+ +	Pre- and through hardened steels**

* austenitization of the steel material; ** note the tempering resistance in comparison to the process temperature

7.2 Thermal Treatment

Thermal treatments include the vacuum, laser, and flame hardening methods, which protect the components against wear through hardening. Currently, the vacuum hardening method is mainly used.

7.2.1 Vacuum Hardening

In vacuum hardening, the part to be treated is heated in a leak-proof vessel. A vacuum is generated by pumping out the air.

Vacuum hardening is suitable for high-strength steels, hot- and cold-work steels, stainless and acid-resistant steels, as well as high speed (HS) steels.

When loading the hardening furnace, one must ensure that only approximately equal workpieces are hardened together. This is because different workpieces have different hardening times. If the hardening furnace is loaded, a vacuum of up to 10 bar is generated by pumping the air.

Depending on the workpiece dimensions, the furnace is ramped up in multiple steps to avoid stresses. For steels with a hardening temperature of more than 900°C, three equalization steps at 400, 600, and 850°C should be applied.

After the long holding time required for hardening the workpieces is reached, the workpieces are cooled, as in standard hardening. In vacuum hardening, cooling is done continuously using high-purity nitrogen at a pressure of up to 6 bar in the furnace area. The rule of thumb for the half-time (thorough heating) is based on 0.5 min. for each millimeter of wall thickness.

The cooling cycle is ramped down to avoid hardening cracks and stresses in workpieces with large cross-section differences. The cooling phase should end with a compensation stage between 150 and 100°C. Lower temperatures unnecessarily increase the stress state.

7.2.2 Laser Hardening

A better, more reproducible surface hardening is achieved by laser hardening. Here, a very specific, controllable hardening to a depth of up to 2 mm can be achieved with a laser beam.

7.2.3 Flame Hardening

This hardening process is done through a partial heating of the surface to hardening temperature and subsequent cooling. In the past, flame hardening was the most commonly used method in mold making. Flame hardening is performed with an oxyfuel torch. The uniformity of torch distance and torch speed has to be ensured.

In thermal hardening processes, the surface of the mold is partially heated and then cooled. Depending on the choice of steel, case hardening depths of about 1 – 5 mm are possible.

7.3 Thermochemical Treatment

In the thermochemical process, carbon, nitrogen, or both are diffused into the steel as it is being heated to hardening temperature. These diffusible substances can be supplied in various forms such as powders, pastes, granules, liquids, or gas mixtures.

7.3.1 Gas Nitriding

In nitriding, nitrogen is incorporated into the surface layers of the steel by diffusion. Depending on the type of steel, hardnesses of about 700 – 1200 Vickers hardness (HV) can be reached. In this process, there is no structural transformation. The process temperatures are between 450 and 580 °C. Depending on the treatment type and layer thickness, an increase in surface roughness can occur. By subsequent polishing, the roughness can be brought back to the initial value.

7.3.2 Plasma Nitriding

In plasma nitriding, plasma is generated in a vacuum by electric charges, the so-called glowing. Here, nitrogen-containing gas is supplied at a voltage of up to 1000 volts, resulting in ionization. Gas ions bombard the molds to some extent. The mold is heated and nitrogen diffuses into the surface of the workpieces. Plasma nitriding minimizes dimensional and shape changes due to the lower process temperatures involved. This is advantageous in long, slender, warpage-sensitive components.

7.3.3 Carburization

During carburizing, also called case hardening, carbon diffuses into the surface layers of the mold steel. Depending on the duration of the carburization process, a layer thickness of up to 2 mm can be established. In this process, a structural change occurs in the carburized layer. The process temperatures in this process are at about 900 °C.

Thermochemical hardening processes increase the surface hardness approximately tenfold above thermal processes. In all the nitriding processes mentioned, no steel warping occurs through the hardening process.

7.4 Electrochemical Treatment

Electrochemical processes are used for wear and corrosion protection of molds. They are preferably used in the plastics processing of aggressive media such as polyvinyl chloride (PVC).

7.4.1 Hard Chrome Plating

Galvanic hard chrome plating is a process that has been used for a long time in the plastics industry to protect against mold abrasion and corrosion. An increased wear and a stronger corrosion resistance occurs, especially in reinforced or filled plastic molding compounds, such as glass or carbon fibers, mineral powder, talcum, or cellulose. All of these additives are abrasive. Therefore, with hard chrome plating, the surface of these molds is protected against wear.

Small- to medium-sized injection molding inserts are through hardened and hard chrome plated as required. Large molds, such as those for car bumpers, are made from tempered steel and then hard chrome plated. Through hard chrome plating a surface hardness of 700 – 1100 HV is achieved.

Prehardened mold steel is used in large molds because of warpage that can occur during hardening. Hard, but also brittle, surfaces tend to develop microcracks. At high cavity pressure, these cracks can lead to chipping.

7.4.2 Chemical Nickel Plating

Another protection, mainly used against corrosion, is chemical nickel plating. This process is used when deep grooves in openings or holes have to be protected against corrosion. This is especially important in the area of cooling cores and dies.

In molds that have to be protected against abrasion and corrosion, both processes can be applied in combination. Nickel plating is done first, followed by chrome plating.

A great advantage of electrochemical processes is the low process temperature of 60–80 °C. Any mold warpage is excluded in this process. A ground and polished surface is prerequisite for high quality plating.

7.5 Chemical Physical Treatment

The hard surface coating is meant by the chemical physical process. Different processes are available for different areas.

7.5.1 Chemical Vapor Deposition (CVD) Coating

In the chemical gas-phase CVD coating, the deposition of solids from the gas phase is done chemically. This method is characterized by a very high wear resistance and an excellent layer connection. Therefore, machining, punching, bending, and compression molds are surface coated with this method. The layer thicknesses are from 2 to 10 microns.

All steels that are common in mold making, such as high speed steels (HSS), sintered metals, and cold work steels, can be coated with this process. Disadvantages are the high process temperatures of up to 1000 °C.

7.5.2 Plasma Assisted Chemical Vapor Deposition (PACVD) Coating

The PACVD coating is a further development of the CVD process, but only needs process temperatures of about 200 °C. Due to the low temperature, no warpage of the components occurs. This is a great advantage for the mold maker and now a well-established process. The layer thicknesses are 1 to 3 microns.

7.5.3 Diamond-Like Carbon (DLC) Coating

DLC coatings are very hard and are becoming increasingly important due to their outstanding wear-, friction-, and corrosion-inhibiting properties. The coating is less brittle than other hard coatings and can have polymer-like properties. In addition, the coating is resistant to many chemicals, and non-toxic.

7.5.4 Physical Vapor Deposition (PVD) Coating

In PVD, the deposition of the solids from the gas phase is done physically. Steels and hard metals at temperatures between 200 and 500 °C can be coated. The materials to be applied, such as chromium, titanium, or aluminum, are evaporated in an electron beam or electric arc. Therefore, undercuts and bore holes are only partially coatable. Today, the PVD process is less frequently used.

The use of hard coatings improves the wear and corrosion resistance in the processing of abrasive plastics.

7.6 Mechanical Treatment

7.6.1 High Speed Cutting (HSC) Milling

The great advantage of the HSC process is that every contour can be generated and the workpieces can be completed in one setting on the five-axis HSC milling machine. Re-clamping is not necessary.

Another positive aspect is that pre-milled and already hardened components can be completed in the HSC milling process. A comparable surface finish to grinding can be achieved with ball mills, which are guided over the contour in intervals of only a few hundredths of a millimeter.

Another important factor is that no significant temperature rise occurs in the workpiece during machining. This is possible because the feed rate is higher than the thermal transfer speed. Thus, a large part of the heat generated during machining is derived from the milling chips.

HSC milling is a very common machining process in cutting technology because the cutting speed and feed rate can be increased by a factor of five to ten over the standard milling process.

7.6.2 Surface Gloss

The presence of a surface gloss is becoming increasingly important. An even distribution of gloss is particulary important in the visible areas of large molded parts.

The gloss of the molded part is essentially dependent on how accurately the mold surface is molded from the plastic.

A rough surface of the cavity is matte with an accurate impression. However, it is glossy with an inaccurate impression. The reason is that the molded part does not fit closely to the mold wall and then cools. The residual heat of the molded part causes a smoothing of the surface.

The relevant factors for the gloss of a molded part surface, besides the quality of the mold surface, are the melt temperature, the injection pressure, and the injection speed.

The surface gloss is not purely physical, but also a physiologically (vision) and psychologically (mood) related variable. The gloss is measured with a reflectometer according to DIN 67530.

7.6.3 High-Gloss Polishing

Even today, polishing is largely carried out by hand. Polishing machines are only used for flat surfaces and round bodies. Machine polishing achieves greater evenness than manually polished surfaces.

The surface roughness after polishing is 0.001 to 0.01 microns. Required are steels that are free from slag inserts and have a uniform structure. Such a structure usually exists for deformed, tempered, semifinished parts.

Copper materials can be polished very well in the hardened state. Pre-polishing begins with felt or sisal polishing wheels with appropriate polishing pastes with a graining of about 10 microns. For the next step, flannel or fiber buffing wheels with correspondingly fine polishes are recommended. If necessary, fine natural hair brushes are used at the end.

7.7 Surface Graining

An increase of the surface finish can be achieved by photochemical etching. Essentially, the technique is based on etching metal through acid. The interaction of the metal, the film structure of the structure image, acid exposure, and time determine the structure of the surface.

The technique of etching is based on the fact that acid attacks the metal and thus etches it away. If acid is poured over the entire mold insert, the entire surface is just etched away deeper. The etching of the steel at this point is prevented by partially covering the steel with a protective coating. The result is a two-level surface structure. The depth of the structure is dependent on the exposure time of the acid.

The desired graining or structures are transferred to the mold surface, similarly to a copying process. A film is produced from the desired surface structure. It is copied onto a film. Using the film, the protective coating is applied to the mold insert surface.

After the etching process, the protective paint is washed off. A surface with the structure of the film is created. The templates for the film are supplied by natural materials, such as leather, wood, cork, or textiles. Multiple etchings refine the structures as desired.

All areas of the molds that do not need to be structured are covered in advance with PVC adhesive tapes or paints. In etching of surface structures, one has to ensure that all mold insert components are made from the same steel to avoid differences in the etching pattern. If a different gloss level can still be seen, a more uniform image can be created by sandblasting. All surface finishing processes, such as nitriding and coating, should be carried out after etching.

Figure 7.2 Molded part with graining (Source: Willi Haller)

Structural etching improves the surface design of the molded part. Structures can refine, decorate, embellish, or laminate. The lamination can make sink marks and weld lines less noticeable.

7.8 Steel Selection

The steel selection criteria include suitability for machining and for eroding and toughness. Another important feature to consider when choosing the type of steel is a good polishability and good photoetching properties. In addition to hardening, movable elements such as slides, ejectors, and stripper plates are also coated to achieve better sliding properties. The powder metallurgy steels are becoming increasingly important.

Table 7.2 Most Common Steel Types in Mold Making

Material-Nº	Steel type	Application
11.730	Non-alloyed	Mold assemblies
12.311	Pre-hardened	Mold assemblies, Mold compounds
12.312	Pre-hardened	Mold assemblies, Mold compounds
12.343	Through-hardened	Highly stressed molds
12.767	Through-hardened	Well polishable
12.083	Corrosion resistant	All thermoplastic + PVC molds
12.764	Case-hardener	High compressive load

7.9 Aluminum Molds

There are many good reasons to use aluminum molds for small and medium batch sizes. There are no size limitation for aluminum in the construction of thermoplastic molds.

Especially for large molds, the advantages of aluminum come into play. A weight reduction of approximately 50 % in aluminum molds can be assumed in spite of a thicker plate thickness and possible ribbing for reinforcing the mold. The machining capacity in aluminum is also about 50 – 70 % higher than steel.

This high machining capacity and the lower weight reduce the throughput times of aluminum molds by about 30 % compared to a steel mold.

Aluminum materials have been specifically modified for mold making. Plates in almost every common size for mold making are available. As far as the basic hardness is concerned, different types of materials are available. One has a choice between casted, forged, or kneaded material plates. These plates have hardnesses of up to 600 kN/mm^2.

Aluminum as a material can be polished. By kneading or forging processes, the density is so high that no pores are visible when polishing.

Aluminum can be machined by wire cutting and electrical discharge machining (EDM). Aluminum molds and their applications can be hard chrome plated, nickel plated, or PVD coated.

If aluminum molds are equipped with hot runner systems, steel rings or similar support elements should be installed on the contact surfaces for support. There are two reasons. The hot runner can be pressed into the aluminum plate and the good thermal conductivity dissipates too much heat to the nozzle. Thus, the nozzle can freeze.

References

1. Krüth, S., Mennig, G., *Mold-Making Handbook* (2013) Hanser Publishers, Munich
2. Vetter, R., Mennig, G., *Mold-Making Handbook* (2013) Hanser Publishers, Munich
3. Johannaber, F., Michaeli, W., *Handbuch Spritzgießen* 2nd edition (2004) Hanser Publishers, Munich

The advantages of aluminum molds are high cutting performance, good thermal conductivity, and a much lower weight.

8 Machining Processes

8.1 Sinking Electric Discharge Machining (EDM)

EDM is a thermal removal manufacturing process for electrically conductive metals.

In sinking EDM, sparks are generated between an electrode and the mold through electric discharge processes.

The electrode is placed in a tank with a nonconducting medium called the dielectric (oil or deionized water) and lowered to the mold until sparks leap. Depending on the current strength, sparking occurs at a gap width of 0.005 – 0.5 mm.

These sparks melt the material into a concentrated point and allow it to evaporate. The removal rate is determined by the frequency, duration, gap width, and polarity of the discharges.

The electrode and mold part must be continuously flushed. The electrode lifts at adjustable intervals and the burn-up is rinsed. The material to be processed determines the material of the electrodes. Copper, graphite, copper-beryllium, and tungsten carbides are common electrode materials.

Table 8.1 Roughness Numbers According to VDI 3400 for Eroded Structures

µm	VDI Norm
0.4	12
0.56	15
0.8	18
1.12	21
1.6	24
2.24	27
3.15	30
4.5	33
6.3	36
9	39
12.5	42
18.0	45

The advantage of sinking EDM is the ability to machine components that have already been hardened. It is possible to produce surfaces with different structures and roughnesses as well as complex geometries. In principle, the electrode burns a negative that resembles the plastic part to be produced.

8.2 Wire EDM

Compared to steel extreme performance improvements can be achieved with the wire EDM.

The drilling of a starting hole is prerequisite for the wire EDM. If necessary, the wire is pulled through this start hole and then through the workpiece using deflection rollers. The wire is only used once. Wire guides provide a straight cut and prevent the wire from swinging.

The cutting process is done in the dielectric. This cools the wire and creates an arc that evaporates the material. The wire can run up to 25 min/m.

A surface roughness of 2 microns can be achieved with repeated trimming.

With the wire EDM, narrow cutting widths and plate thicknesses of 200 mm with high dimensional accuracy are feasible. Breakthroughs can be cut without a radius (R 0.01 mm). Wire materials like brass, copper, tungsten, and steel can be used.

Wire thicknesses of 0.02 – 0.3 mm, and materials such as brass, copper, tungsten, and steel can be used.

Materials for the starting hole EDM are tubes made from brass or copper.

Compared to the scrubbing of steel, wire EDM is six to eight times faster, three to five times faster when finishing, and for fine finishing still twice as fast as in steel processing. Even the subsequent polishing operation requires only about one-third of the time compared to steel.

8.3 Welding

The main applications of the welding process include the compensation of wear due to the operation of the mold and the correction of manufacturing defects.

Every mold maker knows the saying “hammering, welding, soldering frees the mold maker from all his needs.” This may only be understood in a figurative sense today. In mold making, welding processes are mainly used for repairing and correcting the mold.

Build-up welding means to locally coat a component. This can be done with the basic material of the component to be coated or with a foreign material. Material of the same type is usually used in build-up welding within a repair. A component can also be protected against abrasion through the application of a harder material layer. In the latter case, one speaks of armor.

The process variations are as follows:

8.3.1 TIG-Welding

TIG means tungsten inert gas, a mixture of argon, nitrogen, and helium. An electric arc is created between a tungsten electrode (without burn-up) and the workpiece. This produces a molten bath into which a welding rod is introduced by hand. The welding rod then melts.

8.3.2 MIG-Welding

MIG means metal inert gas, a mixture of argon and helium. A wire electrode is passed through the gas nozzle. An electric arc is created between the wire tip and the workpiece. The arc can then melt the wire.

8.3.3 MAG-Welding

MAG means metal active gas mixture of argon and helium mixed with carbon dioxide or oxygen. The welding wire, which is wound on a coil, contains both the filler material as well as the current-carrying electrode. The gas, which flows through the welding point, prevents oxidation. The interplay of active gas mixture, arc, and welding additive causes a good deposition rate and low slagging.

8.3.4 Laser Beam Welding

Laser beam welding offers the advantage of the very high energy density of the laser and precise machining. In addition, little or no warpage occurs due to the low heat input.

In the material application in the edge region of contouring molded parts, the rework can be greatly reduced by very fine applications. Portable laser systems allow small repairs directly in the clamped mold.

8.4 Punching und Drop-Forging

In these two methods, a punch, which represents the positive of a molded part to be manufactured, is driven into a metal block. Both methods are hardly ever used anymore.

8.4.1 Punching

Lowering the surface of a metal with a punch is a cold forming process that occurs at room temperature. A hardened punch is pressed under high pressure into a blank of steel, which has a high lead content to increase its toughness. In this way, several identical dies can be produced with a punch.

Cold forming achieves the same dimensional accuracy as EDM. The surface quality is comparable to that of a polished die.

8.4.2 Drop Forging

In drop forging, a metal piece, which is heated to 950 – 1200 °C, is deformed between two mold halves (the die and the embossing stamp).

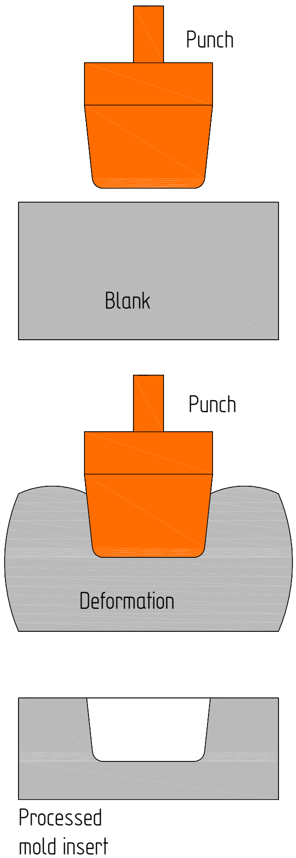

Figure 8.1 Drop forging with a punch

8.5 Casting

Casting is a process in which a castable material is casted into a cavity without pressure. The cavity represents a negative of the article to be produced.

All casting molds are equipped with a gate and a vent. The characteristic of continuous casting molds is that they can continuously be used.

For small series, these molds can be made of wood, gypsum, and now also silicone.

For large series, continuous molds are manufactured from steel.

With casting molds, plastic parts can be produced that are unique because of their model character or are only produced in small quantities.

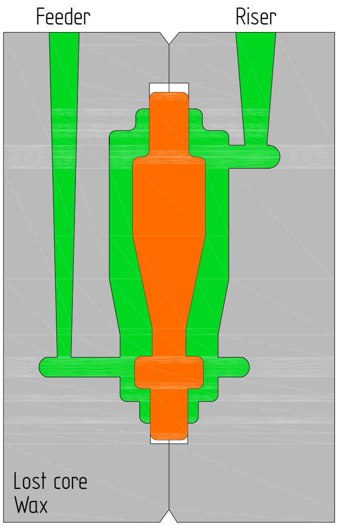

Figure 8.2 Principle of the casting technology

8.6 Galvanized Mold Inserts

Galvanized molds have the advantage that they are based on a positive model, which is easy to manufacture. A corresponding negative galvanized mold insert can then be created.

The positive model can be made from metal or plastic material. However, it has to be made according to the guidelines of electroforming. Complex geometries can also be electroplated very well.

First, a conductive paint is applied to the positive model. This is usually done by a chemical silver-plating technique. Then the model is immersed in a bath that contains an electrolyte consisting of a nickel salt solution.

The positive model acts as a cathode. Nickel plates are hung in the salt solution bath as an anode. The anode and cathode are now connected to a direct current circuit. The current causes nickel ions to flow from the anode and be deposited at the cathode. After 4 – 5 hours, a nickel layer with a thickness of 3 – 4 mm is formed on the positive model. Frequently a combination of nickel with a subsequent copper layer is applied. Depending on the application purpose, a wall thickness of up to 20 mm can be created.

Galvanized mold inserts with a Rockwell hardness of 44 – 48 HRc are usually used. Perfectly manufactured nickel layers are nonporous and have an excellent surface quality. Using electroforming, the finest contours of the positive model surface can be exactly transferred. Real leather graining, the finest diamond cuts, and skin structures are even possible.

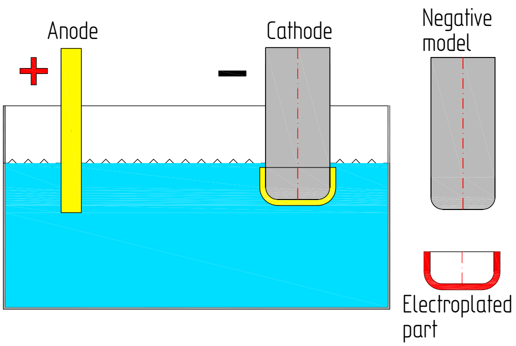

Figure 8.3 Model of an electroplating facility

> Galvanically produced mold inserts are an affordable alternative to traditional production using erosive procedures, including EDM. The surface structures of the positive model are exactly reproduced in the negative mold insert up to the micro range.

9 Care, Maintenance, and Storage

9.1 Mold Care

Regular care of molds pays off in the long run. Only a few simple steps are needed daily to ensure the long-term production safety of the molds.

Regular mold care may be as follows: A visual inspection should be performed during change of shifts or before the start of production. Here, it is important to check the parting planes, the sliders, and the guide elements. Possible deposits and plastic fluff or threads should be removed. The moving components, such as sliders, should be checked for free movement, and the slider paths should be cleaned and greased if necessary.

If the injection molding machines are switched off after the end of the shift, the mold must be opened and the parting planes cleaned with a towel and blown out. Then, the parting planes need to be sprayed with preservative oil and the machine closed to within a few millimeters to prevent condensation from forming between the parting planes. Also, if the injection molding machine is to be unused for several days, it is recommended to blow out the cooling system. This prevents corrosion damage due to the cooling fluid. If a rust-protective agent is added to the cooling agent, blowing out can be eliminated.

If a mold is taken out of production, the care must be a bit more extensive. It is advantageous to perform this at a special workplace where molds can be opened and closed. If a workplace like this is not available, the care can be handled on the injection molding machine before removing the mold. This variant is more expensive because of the high hourly machine rate.

Care Procedure

The mold is opened, and all moving elements are removed and cleaned. Hot runner systems should be checked for leaks, and sprue bushings should be checked for damage at the radius of the machine nozzle. When all the cleaning and maintenance steps are completed, the mold is sprayed with a preservative (see Section 9.2) and deposited in the mold storage area.

It is advisable to create specification or maintenance lists for recurring maintenance measures. Many mold makers deliver these in conjunction with a spare parts package.

9.2 Inspection

The molds should be preserved after the end of production. It is important to note that only silicone and Teflon-free preservatives should be used.

The molds should cool down before storing so that no condensation occurs on the mold surface, which can lead to corrosion.

The temperature control channels should be closed. To do so, self-sealing water couplings can be used. It is also advisable to flood the cooling bore holes and channels with a liquid with preservatives to prevent the ingress of oxygen.

For the preservation of sliders, cores, and other components, the complete spraying of the elements with gun oil has been proven successful. The oil creeps into the smallest gaps and is well distributed on the surface of the mold and the components. Gun oil also adheres well, removes carbon deposits, and keeps the mechanism lubricated.

Molds on which hard or soft polyvinyl chloride (PVC) has been processed must be particularly well cleaned and preserved in the gate area. Hydrochloric acid is released in the processing of PVC, and the mold has to be protected against it.

Once all important items of the preservation are completed, the mold can be stored. One has to pay attention that the mold is not completely closed on the way to storage. Transport bridges with a minimal opening of 5 mm between the parting planes are helpful. This ensures that any condensation can drain and the mold can dry again.

It also makes sense to perform a thorough inspection of associated additional equipment, such as grippers for handling devices, clamping and control gauges, as well as spares or replacement parts.

The inspection should be performed within a fixed period of time and according to a specific inspection plan for each mold. A regular and thorough inspection usually obviates unplanned maintenance during production.

9.3 Maintenance

In general, molds should be checked and repaired, if necessary, before storage.

For small- and medium-sized businesses, maintenance is carried out during mold making. These businesses usually have their own repair department where maintenance, which is specified by the mold suppliers in the requirement specification, is performed and worn parts are replaced, if necessary.

In addition, the connections of the hot runner systems, end switches, and other monitor equipment should be checked. The recording of all damages is helpful to draw conclusions on possible design errors. A properly operated error analysis helps to identify weak points and thus avoid interference.

In maintenance, molds that are coming out of production or going into production are treated and prepared. If maintenance work is due during production, the machine operators generally perform it themselves.

It has proven useful in practice to record the last shot in multi-cavity molds. Small errors in individual cavities can therefore be identified easily, can be assigned to the cavity, and repaired before storage.

9.4 Storage

Molds should be stored dry and protected against fire. They should also be maintained before storing so that they are able to start quickly at the beginning of production.

Injection molds must not be stored together with raw materials, fats, and oils. Central mold storage some distance from the production area is ideal. It is also advisable to store additional equipment, such as robotic grippers and special equipment that are tailored to the respective mold.

The general rule is that all devices that have to be replaced in the short term should be specially protected against fire and extinguishing water. Fire doors and fire walls are key additions for the safe storage of molds.

In addition to identifying the storage locations, the mold identification must be attached on the shelf. The following information is useful for the assignment of the molds:

- Mold number
- Part identification
- Measurements
- Weight
- Manufacturer
- Year of manufacture

For rapid identification, it is recommended to attach a molded part to the mold.

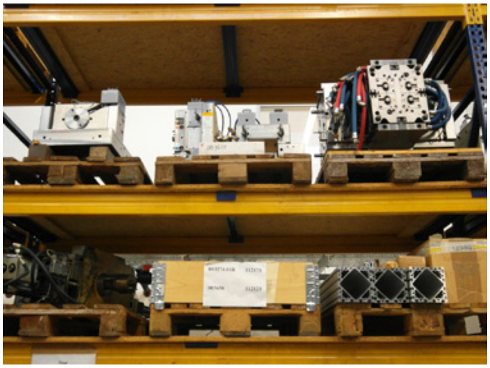

Figure 9.1 Storage of molds on heavy-duty shelving (Source: FOBOHA)

Storing the molds on heavy-duty shelves is recommended. The storage locations should be marked so that they can be assigned to the molds and are easy to find.

Index

U

V

W