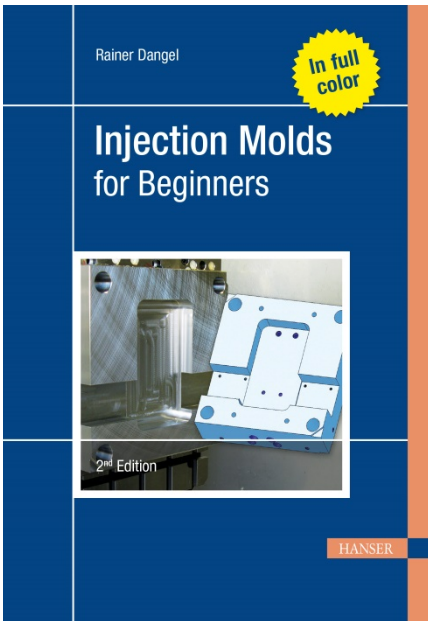
HANSER



Sample Pages

Injection Molds for Beginners

Rainer Dangel

ISBN (Book): 978-1-56990-818-1
ISBN (E-Book): 978-1-56990-819-8

For further information and order see

www.hanserpublications.com (in the Americas)

www.hanser-fachbuch.de (outside the Americas)

Preface to the Second Edition

First of all I would like to thank my readers warmly. The success of this work has shown that its creation was a valuable exercise. The extensive feedback I received was consistently positive. The English edition, like the German edition, has been well received, and it has been a pleasure to see that it has been sold throughout the world, including in China and India, where it has been actively used in companies and training institutes. The popularity of the book is also the reason why the second edition is now available in full color.

Of course I was asked several times how a mold maker came up with the idea to write such a book. How does he find the time and where does the comprehensive knowledge come from?

The motivation to write a book can be manifold. My motivation was to write a small manual for the distribution of machining centers for mold making. The sales department should understand what mold making is, what it does, which components are to be manufactured, and from which materials the individual components are made. At first I wanted to use existing documents and publications. But I came to realize that there was nothing suitable at this level for beginners or newcomers. Then the only thing left was to create something of my own.

In order to make the book understandable for everyone, the idea came to me to always use the same plastic part as the basic concept. It should be as simple as possible and concentrate on the essentials. This gave me the possibility to build up the level of difficulty of the plastic part stepwise, and to explain the thereby-arising changes simply. That is to say, the central thread throughout the whole book should be uncomplicated and understandable. After publication in the processing machine company, the books were all gone after a few days. Not only the sales department, but also other interested parties tried to get hold of one.

So what could have been more obvious than to create a large work from this small book? Especially since, as already mentioned, there was nothing comparable on the market. At first, time constraints meant that the project ran more and more behind schedule. Then a long serious illness brought me the time, which I then used. Over 2500 hours of work and about 40 designs, or revisions of designs, had to be accomplished. Including all the corrections, this project took considerably more than half a year. The result is the present book, which after its success in both German and English versions, is now available in a second edition.

Over 40 active years in mold making, more than 23 years of which I worked as an independent entrepreneur and now as a project manager, consultant, and instructor have brought the necessary knowledge and experience. My training began in the summer of 1976. I passed through the entire technological change from milling machines with a handwheel, to NC technology, and then to today's 5-axis simultaneous machining. The first designs were produced with India ink on a drawing board, moving on via a simple 2-D CAD program already in 1995 to full 3-D CAD.

The change over the decades was not only in the technology of the production of the injection molds, but also in the necessary shift from a handicraft business to an industrial company. Today the customers of the mold maker are almost exclusively industrial companies. Certifications, creation of processes, and Industry 4.0 are keywords that have occupied the mold making industry in recent years.

This is also the reason why a new section on process chains has been included in this second edition. In addition, it has been technically extended and small errors that unfortunately crept into the first version have now also been eliminated.

I hope you enjoy reading this book and look forward to your feedback.

Rainer Dangel, February 2020

Foreword to the First Edition

German die and mold making is a brand with global significance. The reasons for this are diverse, but the industry's secrets to success can certainly be attributed to smart design with a great deal of know-how, top performance production engineering and quality related criteria. One major aim of this book is to disseminate this philosophy to a wider, English-speaking readership.

Rapid implementation of innovations through close information exchange between all parties is planned for the future. Injection molds today already play a key role in modern production engineering in the manufacturing industry. Visions of the future such as the "smart factory" in the context of injection molding now offer the chance to raise the energy and resource efficiency of the production process to a new level with intelligent management and network flexibility. But the basis for this is a solid knowledge of the basics of engineering and manufacturing processes in mold making. The above-mentioned topics can only be implemented based on this knowledge and wealth of experience. And this is exactly where this technical book from Rainer Dangel comes in. What is required for bringing a product into shape?

In the book the author didactically as well as technically breaks new ground in the field of technical literature for injection mold making. In a very clear way, he combines theory with practice, always focusing on the following questions: "What is this product relevant for? What needs to be solved technically for which product specifications?" And, regarding the method of the manufacturing implementation: "How and with what can I fulfil the product requirement within the scope of the design and also the manufacturing process?" Through Mr. Dangel's technical expertise which he established and developed over many years, it quickly becomes clear when studying the book that the practical implementation of the described has great significance. Basic knowledge and solutions are holistically considered. Advantages and disadvantages are presented and discussed. The wealth of 35 years of experience, beginning with training as a tool maker to the master craftsman's diploma then to owning a private company flows through this technical book.

“Injection Molds for Beginners”, the title of this book, hits the bull’s eye and old hands who think it is no challenge to them might be taught a lesson!

Prof. Dr.-Ing. Thomas Seul

Vice rector for Research and Transfer at the Schmalkalden University of Applied Sciences and President of the Association of German Tool and Mold Makers (VDWF).

The Author

Rainer Dangel began his professional career in mold making with training as a mold maker from 1976 to 1980. As a young skilled worker, he already realized the possibilities of making a difference in this emerging technical profession. He laid the foundations for his career with the master craftsman's diploma in mechanics at the age of 23.

He segued into self employment in 1987. He began with a small CNC milling shop for mold making components which within a few years developed into a modern, technically high-quality specialist company for manufacturing injection molds for various requirements. He had already introduced and was using the first 3-D CAD CAM system successfully in 1995.

All manufacturing options of modern mold making could now be offered. Rainer Dangel made it his duty to actively continue to develop and perfect the manufacturing processes. In 2006, the company built their own injection mold making shop in order to expand the process chain and be able to supply finished plastic components. Through the certification in accordance with DIN EN ISO 9001:2008 in 2008, his company was able to supply a variety of industries. Among other things plastic parts for the automotive industry could be VDA tested and approved (VDA = Association of German Automobile Manufacturers, see *https://www.vda.de/en*).

In the generally difficult economic year of 2010, the mold making company was closed. He was then for several years head of the technology center at the Heller brothers machining company in Nürtingen, Germany, where he was responsible for the support of the customers in the area of die and mold making.

Now back in the mold making arena, Rainer Dangel is currently active in two main areas as both consultant and instructor: Firstly, project management for injection molds, from part design via mold design all the way to start of production; he is also a machining specialist. The second area is the education and training of young people in this industry. He holds a lectureship at Reutlingen University, Germany, and supervises mechanical engineering students specializing in plastics technology.

Acknowledgments

I would like to express a heartfelt thank you to my colleagues at the Association of German Tool and Mold Makers (VDFW) for the support during the development of this book. Special thanks to Prof. Dr.-Ing. Thomas Seul, President of VDWF, for the foreword.

- Formenbau Schweiger GmbH & Co. KG, Uffing am Staffelsee, Germany, Anton Schweiger (Vice President)
- Formenbau Rapp GmbH, Löchgau, Germany, Markus Bay (Director of Training)
- VDWF, Schwendi, Germany, Ralf Dürrwächter (Managing Director)
- bkl-Lasertechnik, Rödental, Germany, Bernd Klötzer
- exeron GmbH, Oberndorf, Germany, Udo Baur
- Gebr. Heller Maschinenfabrik GmbH, Nürtingen, Germany, Marcus Kurringer, Jörg Bauknecht
- GF Machining Solutions GmbH, Schorndorf, Germany, Gabriele Urhahn
- Hans Knecht GmbH, Reutlingen, Germany, Hans Knecht
- Reichle GmbH, Gravier- und Laserschweißzentrum, Bissingen, Germany, Volker Reichle, Marco Reichle
- Werz Vakuum-Wärmebehandlung GmbH, Gammertingen-Harthausen, Germany, Henry Werz
- AS-Beratungen, Amtzell, Germany, Andreas Sutter

- 3D Systems GmbH, Ettlingen, Germany (previously Cimatron GmbH)
- Meusburger Deutschland GmbH, Viernheim, Germany (previously PSG Plastic Service GmbH)
- Meusburger Georg GmbH & Co KG, Wolfurt, Austria
- MAKINO Europe GmbH, Kirchheim-Teck, Germany

The following are not association members, but were also on hand to help me. For this a heartfelt thank you to:

- Friedrich Heibel GmbH Formplast, Heuchlingen, Stefan Heibel
- Carl Hanser Publishers, Munich, Ulrike Wittmann, Jörg Strohbach

Finally, I would like to thank the Translation Management department at Meusburger Georg GmbH & Co. KG, Wolfurt, Austria, in particular Kristin Bylund Thurnher and Birgit Lins, for their expert translation of my German text into English. Achieving a high-quality translation of a specialist technical book is no trivial task, and for this the professional support of Meusburger is most warmly acknowledged.

Contents

How to Use This Book

In this book the planning, designing, and construction of injection molds is explained and described. It deals exclusively with injection molds for thermoplastics processing.

To simplify matters, the term “injection mold” is also referred to as mold, but has the same meaning. The term mold established itself in the specialist world and is predominantly used there. Note also that the spelling “mould” is used in British English, but again the meaning is the same.

Everything is explained and described concretely and understandably. A plastic container with a cover is the basis for almost all explanations. The drawings and designs of both of these plastic parts were especially made for this book. The dimensions of the designed molds and the technical details are real, so the injection molds can be actually built. On the basis of both or one of these parts, as much as possible is shown and explained.

There are sample calculations for the planning and dimensioning of injection molds. Different functions and elements relevant for the design are explained in detail. With the increasing demands on technology in the mold, the two parts become ever more complex so there is always a reference to the previous topics. If the part and/or the mold becomes more complex, the reason for it is therefore comprehensible.

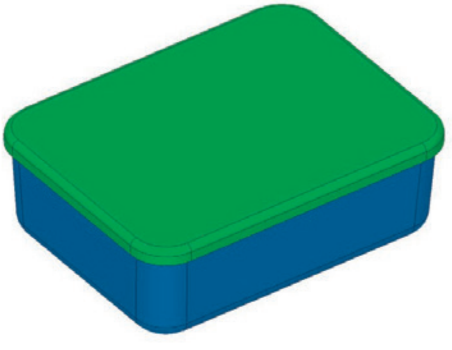


Figure 1 Container with cover

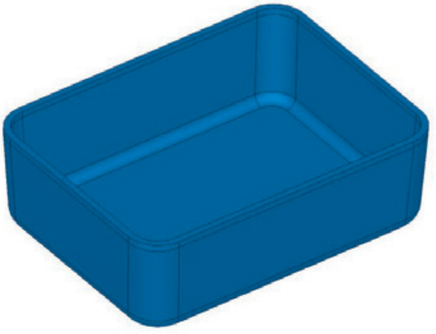


Figure 2 Container

Figure 3 Cover

There are further chapters in which the existing designs of actually manufactured injection molds are the basis for the explanations.

2 Mold Types

2.1 Simple Open/Close Mold

The open/close mold got its name from its easy movement and function when the injection mold for machining of the plastic parts is clamped onto an injection molding machine. The injection mold or the injection molding machine opens and closes without any further necessary movement taking place in the injection mold.

The entire motion sequence is called an injection cycle or just cycle. It begins with a closing of the injection mold. When it is closed, a liquid, hot plastic mass is injected into the injection mold under pressure. Now a certain amount of time must pass before the liquid plastic has cooled and solidified and the plastic part in the injection mold reaches a certain stability. The injection mold opens and the finished, still-warm plastic parts are ejected from the injection mold. When all of the movements are finished, the process starts again. For the outside observer, the machine opens and closes again and again.

> In using the term “liquid plastic”, one is referring to plasticized plastic. Plastic pellets are heated and plasticized, which means they become soft and capable of flowing. In this consistency, the plastic can be injected into the injection mold. Depending on the type and kind of plastic pellets, this vary from being highly viscous to having a water-like viscosity.

The direction in which the injection mold or the injection molding machine opens and closes is called the main demolding direction. All movements of the injection molding machine, the injection molds and the moving parts in the injection mold run in this axial direction. Depending on the component there can be additional demolding directions. This is described in Section 2.2.

The open/close mold is the simplest of all injection molds. As a result it is often the cheapest. Already in the planning and designing of plastic parts, efforts are made so that the plastic piece can be produced with this type of injection mold.

Figure 2.1 shows the demolding direction of a simple open/close mold. Both upper part (fixed half) and lower part (moving half) open and close in an axial direction. The plastic part has been designed for being produced with this specific mold in such a way that when opening the mold on the injection molding machine it is not damaged or destroyed.

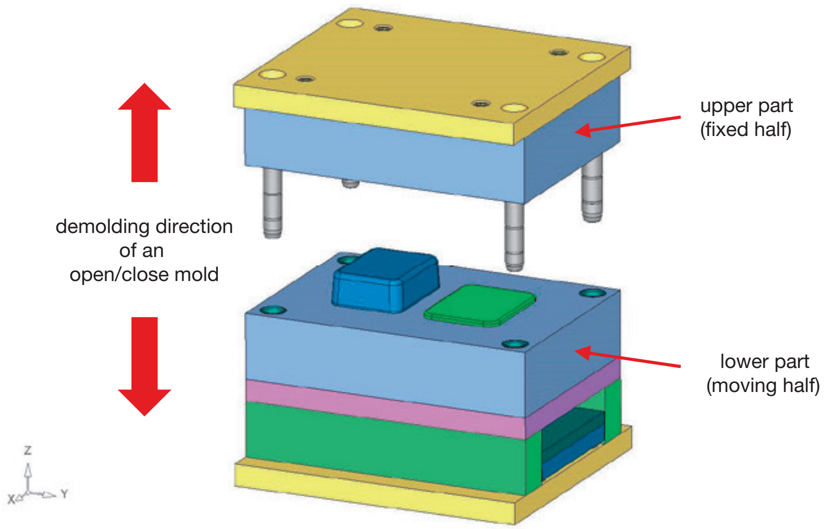


Figure 2.1 Demolding direction

The plastic parts which are to be produced with such an injection mold have no structural elements which deviate from the main demolding direction. Cup-shaped or flat parts, for example, are manufactured with this type of mold.

A plastic part can have elements such as side openings, latches and clips, laterally protruding edges or pipes. For the demolding of these elements, moving components—called slides or inserts—are designed for the mold. In a secondary demolding direction, these elements called undercuts can be removed from the mold without damage. More on this in Section 2.2.

2.2 Molds with Moving Elements

Almost everything that makes an injection mold complicated and expensive originates from the geometry of the subsequent plastic parts. Therefore attention should already be paid in the planning and design of this plastic part that everything that should later contain the plastic part is also to be realized in the injection mold. This is often a big challenge in the development, that is, the design of plastic parts. When design and technology meet, sometimes one has to compromise.

2.2.1 Undercut

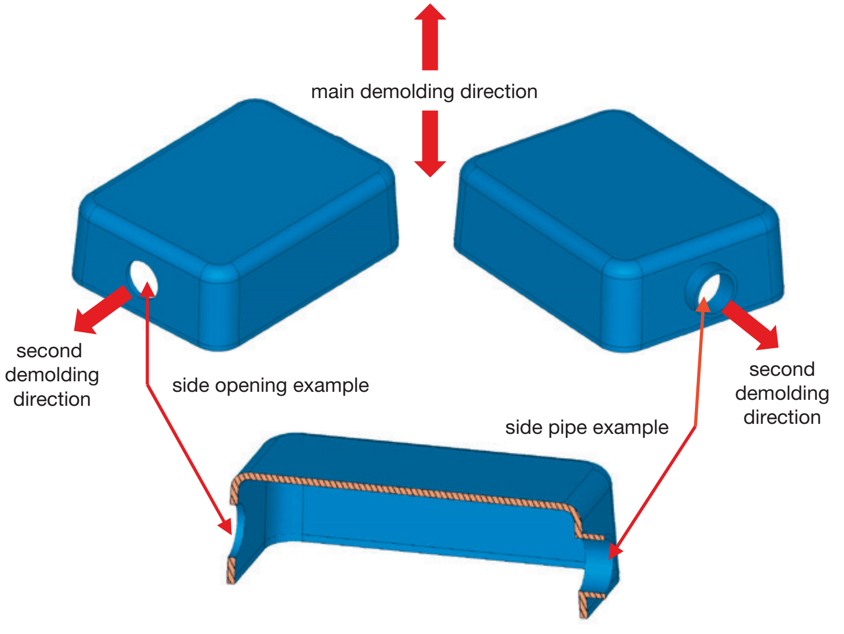


Figure 2.10 Additional demolding directions

The next level of difficulty in plastic parts is elements which cannot be demolded in the main demolding direction like in an open/close mold. These elements, which are troublesome during demolding, are called undercuts. They need to be released or demolded in an additional demolding direction. For this purpose moveable components, such as slides, core pins, ejectors for inclined ejection units or inserts, are used in the injection mold. They support the plastic piece so that it can be better demolded and ejected.

In Figure 2.10 two possible elements, a side bore hole and a side pipe, are seen on our component. Both elements are an undercut on the plastic part and must be released via the second demolding direction. Only this way can the plastic parts be ejected from the mold without damage. For these two examples slides are used to do this.

2.2.2 Slide

When implementing these side openings the open/close mold becomes a mold with slides. Slides are moving components inside the injection mold. One or more parts of the mold contour are incorporated into these slides. The slide itself moves away from the plastic part during or after the opening of the mold in an additional demolding direction. Through this movement the undercuts are released before the plastic part is ejected from the injection mold. The required path is calculated and defined in advance. It must be large enough so that the plastic piece drops out of or can be removed from the injection mold without damage after the ejection.

In Figure 2.11 the slide for demolding the side opening on our container is shown. In the front area of the slide a part of the mold contour of the plastic part is incorporated. The round surface in front has contact with the fixed insert when the mold is closed and is injected. During injection, this contact prevents that the plastic covers this spot and thus forms the bore holes in the plastic part. In technical language, this contact point is also called an aperture.

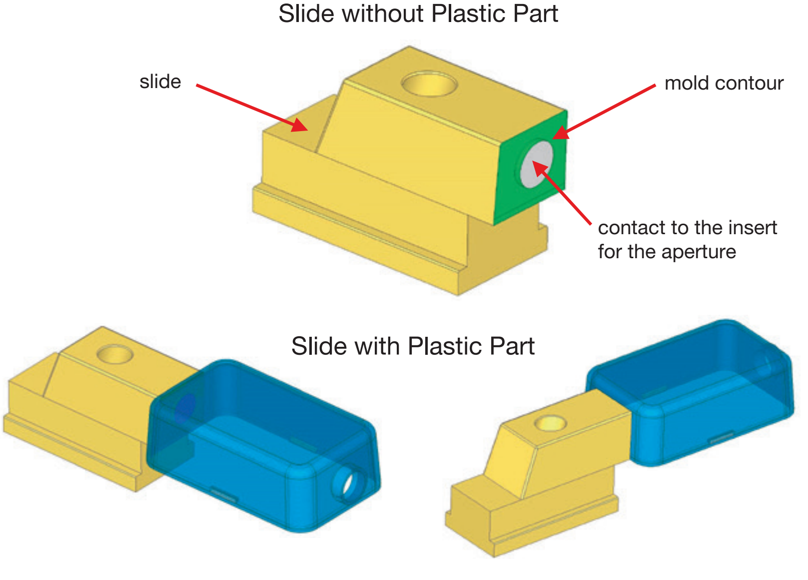


Figure 2.11 Slide with and without plastic part

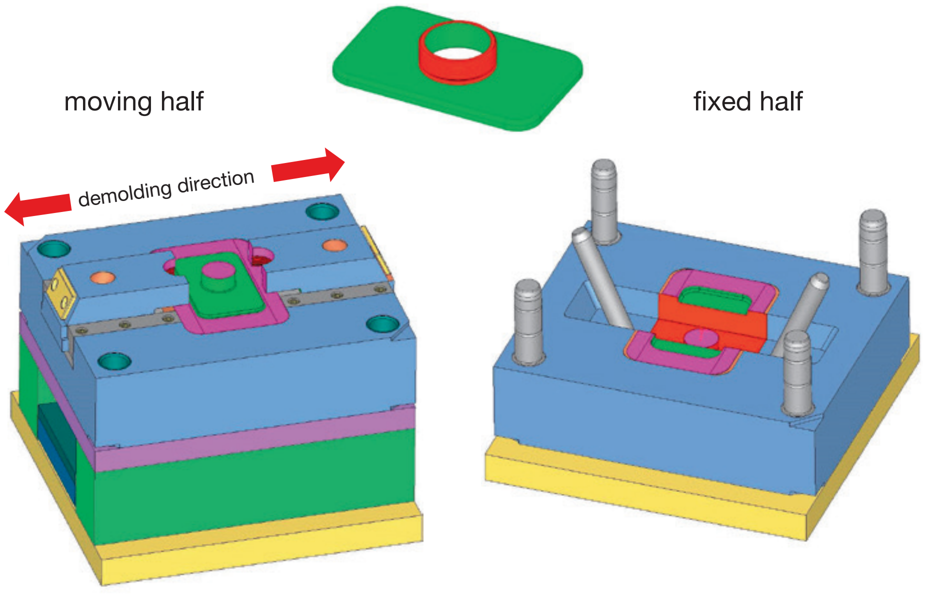


Figure 2.20 Mold for cover with external thread

In Figure 2.21 the slide with all functional surfaces is displayed. Right in the front there is the split line surface in the middle of the injection mold, where both slides and the mold contour—where the thread geometry belongs—meet. In addition, the inclined hole is shown in which the inclined pin for moving the slide is immersed. On the sides the slide guides, in which the slides are embedded and are driven in the demolding direction, are displayed.

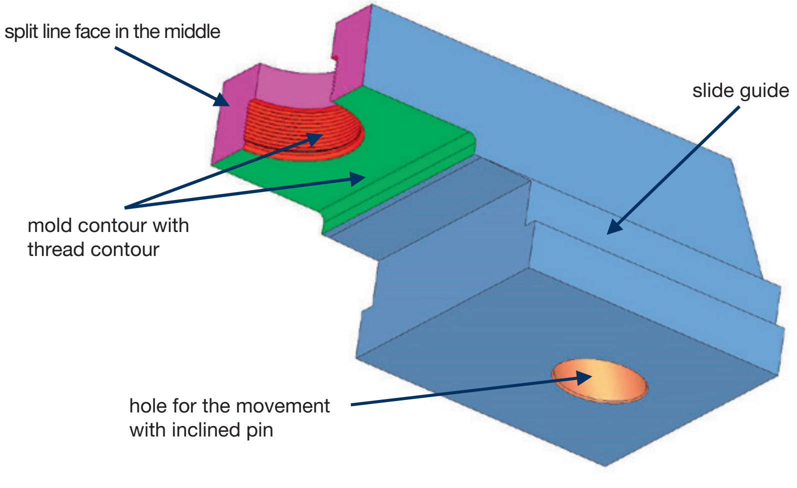


Figure 2.21 Slide for the cover with external thread

Only exception: the contour of the thread *lies* in the mold and is right in the main demolding direction. The threaded plug in the top left corner of Figure 2.19 is such a plastic part. It is produced with an open/close mold.

2.3.2 Internal Threads

The injection mold for the internal thread also belongs to the category molds with moving elements but is a completely different mold concept. The thread itself is also an undercut for this injection mold, but this time it cannot be demolded with slides.

There are two significant options for the demolding of these internal threads:

- Option 1 is the collapsible core. This is explained in detail later in Section 4.3.6.
- Option 2 is the unscrewing of the thread. Specialists call this *de-spindling*. Here the mold core, where the thread is incorporated on the front, unscrews from the plastic part before it is ejected.

The difficulty when demolding the internal thread is that there is no room to move the slide inwardly with a linear movement such as with external thread. The question is, how do you release the thread or undercut? The solution is to unscrew the core from the mold contour using the thread.

Previously, all movements that were important for demolding were linear movements in either the main demolding direction of the machine or the secondary demolding direction through a moving slide. But for unscrewing, a circular or screwing movement is required.

The screwing movement for a core can be produced in different ways. The three common types of operating a mold core are: the unscrewing unit, the drive via a gear with gear rack or a gear with a high-helix lead screw.

> For all types of de-spindling, it must be ensured that the plastic part does not rotate during de-spindling with the turning of the mold core. Often the parts that have an internal thread are round parts. Through serrations on the outer surface or a polygon or small catches on the bottom edge, the twisting can be prevented. This is not a problem for our rectangular cover.

In Figure 2.22 the plastic part is shown with the mold core which is injection molded with the thread.

2.5.2 Hot Runner

The stack mold can only be used with a hot runner (see Section 3.9.10).

We go on with the first variation in which all parts are made from the same plastic. The first hot runner is in the mold on the half which faces the sprue bush of the injection molding machine (moving half 2 in Figure 2.35). This hot runner is only the feeder for the plastic. In the middle section the hot runner with the distributor which fills the entire cavity with plastic is located. In the split line between fixed half 2 and the moving half 2 (see Figure 2.35) there is a transfer point for the plastic. Through the opening and closing of the mold this transfer point between the two hot runners is separated and connected again once in every cycle.

In Figure 2.36, a hot runner system has been integrated in the mold. The transfer point can be seen in the split line 2 (see Figure 2.35). The plastic is injected centrally through the machine nozzle.

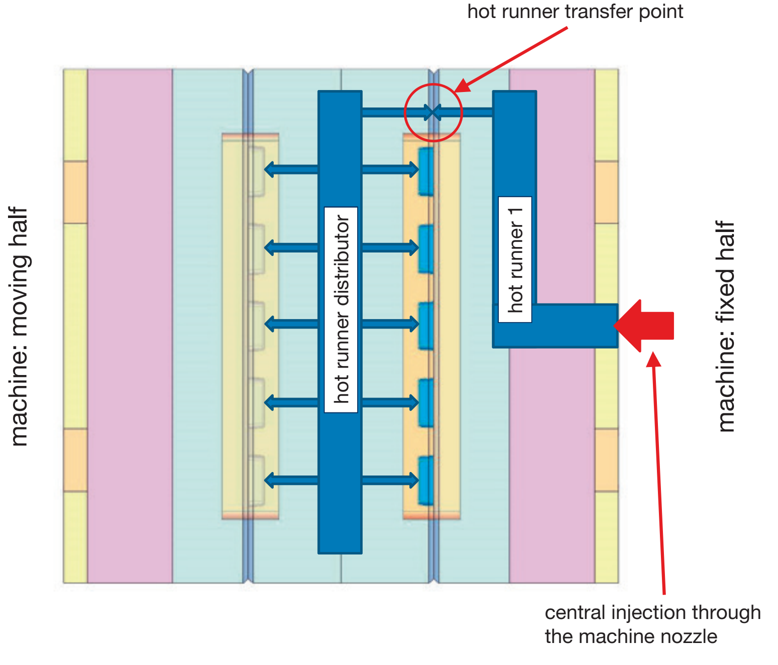


Figure 2.36 Stack mold with hot runner

2.5.3 Opening and Closing

The mold is clamped as usual to the injection molding machine: with a clamping plate on the left to the moving half machine plate and with a clamping plate on the right on the fixed half machine plate. The central part of the mold is held in the center by the guiding between the two moving halves. For this special case, in some machines there are sliding guides on the machine base and the central part is supported with a guide shoe.

In Figure 2.37 the injection mold is clamped on the machine. The central part is supported on both sides by the guide shoe.

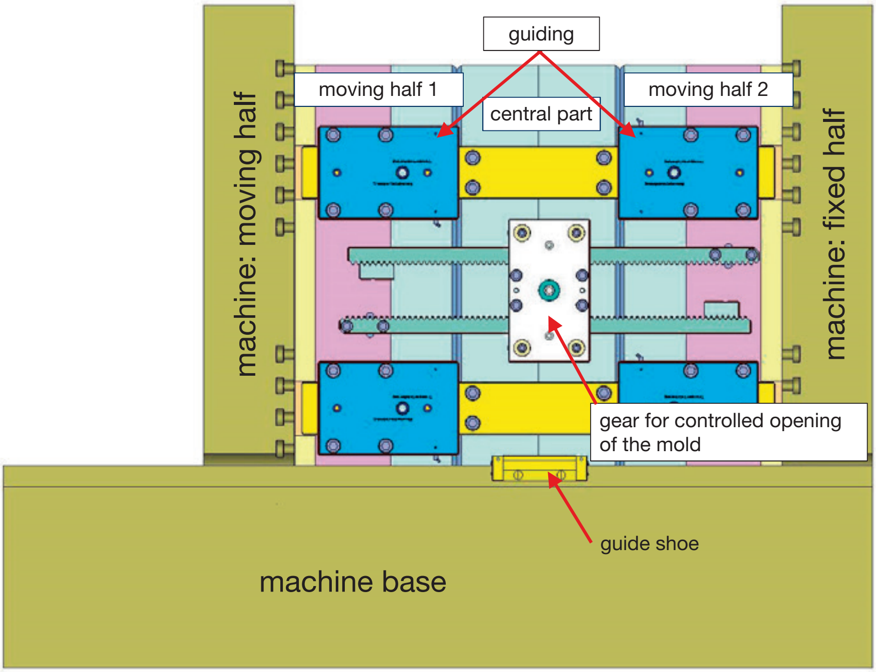


Figure 2.37 Injection mold installed on the machine

When opening and closing, the machine or the injection mold covers a certain distance, in our example 400 mm. The central part is controlled through the gear and travels half the distance, that is, 200 mm, and the opening width is the same for both split lines.

Figure 2.38 illustrates the situation for an open mold. The machine has travelled 400 mm and the opening width of each split line is 200 mm. Therefore there is enough space for the finished plastic pieces to fall from the mold.

quantities? Then mostly only one piece comes out of the mold. The costs for the mold should therefore be kept as low as possible. An example of this is an exclusive sports car which is limited to only 1,000 pieces.

However, if large quantities are required then the number of cavities increases. In such cases 2-cavity and 4-cavity molds are often used. For mass produced goods such as screw caps for drink bottles, the number of cavities is sometimes over 100. For such an application also a stack mold can be used since it enables the production of a high number of parts with just one cycle.

Here for example in the automotive industry the molds with different parts are used. A possible solution is to produce a left and right plastic part each in one mold of a 1+1-cavity or a 2+2-cavity injection mold respectively. The production is done in pairs, for example the right and left external mirrors for a car.

Complexity of the Plastic Part

The more difficult and more complex the plastic piece is, the more costly and expensive the injection mold is. This factor is often considered together with the output quantity. For very complex plastic pieces, only a 1-cavity or if mirrored a 1+1-cavity mold is constructed. An example of such a mold is the multi-component injection mold mentioned in Section 2.4. Although the planned quantity over long periods is very high, the 2+2-cavity mold is the economically best choice here.

Machine Size

Another but very special issue is the machine size of the injection mold which the customer will later produce the plastic parts on. The machine size, whether it is about the clamping surface or the closing force, already limits the size of the injection mold and therefore also the number of cavities.

Mix from Different Plastic Parts

Especially for sample parts or prototypes there are also injection molds in which two, three, or even four different parts from the same plastic are produced. A variant in which money can be saved is when the parts have simple geometry and can be easily integrated in the mold design. The costs for the entire mold base are allocated to all the parts.

Overall, the determination of the number of cavities is a mix of output quantity, complexity and the machine size.

Examples of an Economic Efficiency Calculation:

Requirements for a plastic part over the entire service life of 100,000 pieces.

- Cost for the 1-cavity mold: $20,000. The price per part produced with the 1-cavity mold is $0.50/piece. 100,000 pieces × $0.50 results in an overall cost for the parts of $50,000. Together with the mold costs, a total of $70,000 is calculated over the entire service life.

- In contrast, the 2-cavity mold costs $30,000 but because two parts instead of one part comes out of each machine in every cycle the price per part is only $0.30/piece. If you sum up the costs for the 2-cavity mold, the overall cost for the parts is $30,000 and the overall cost for the mold is $30,000, which together makes $60,000.

  Result: This calculation shows that with a 2-cavity injection mold, a total of $10,000 can be saved over the entire service life.

In Figure 3.18 two examples of molds are displayed: the 2-C mold and the stack mold, comparing their complexity and the output quality.

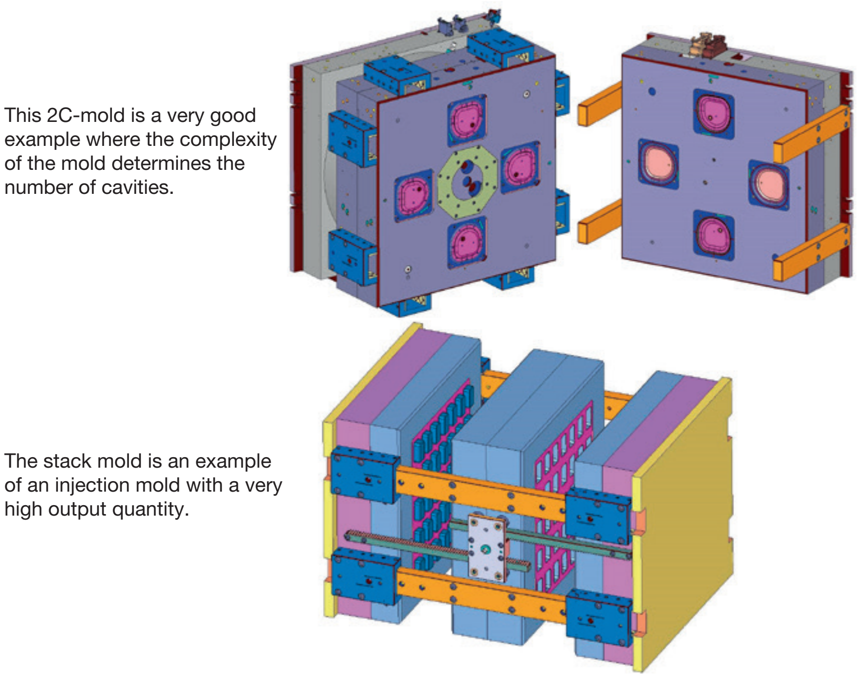


Figure 3.18 2-C mold and stack mold

3.3.3 Arrangement of Cavities

Once the position and number of the cavities are defined, how are they arranged in the injection mold?

Table 3.2 The following tables contain material specifications for injection molds. [Source: Meusburger GmbH, Wolfurt]

3.3547 (AW-5083)	**DIN:** AlMg 4.5 Mn **EN:** ISO 5083 **AFNOR:** A-G4.5MC **UNI:** 7790	Si - 0.40 Fe - 0.40 Cu - 0.10 Mn - 0.70 Mg - 4.40 Cr - 0.15 Zn - 0.25 Ti - 0.15	≤ 290 N/mm² (depending on thickness)	**Aluminum alloy**	Plates for mold bases and jigs
3.4365 (AW-7075)	**DIN:** AlZnMgCu 1.5 **EN:** ISO 7075 **AFNOR:** A-Z5GU **UNI:** 9007/2	Si - 0.40 Fe - 0.50 Cu - 1.60 Mn - 0.30 Mg - 2.40 Cr - 0.23 Zn - 5.60 Ti - 0.20	≤ 540 N/mm² (depending on thickness)	**Aluminum zinc alloy** high-strength, hardened	Plates for mold tools and dies with increased requirements on strength

If the mold is designed for an average quantity of for example 100,000 pieces, a better quality of material is required. For example, the mold frame is made from pre-toughened tool steel 1.2312.

1.2312	**DIN:** 40 CrMnMoS 86 **AFNOR:** 40 CMD 8.S **AISI:** P20+S	C - 0.40 Si - 0.40 Mn - 1.50 Cr - 1.90 Mo - 0.20 S - 0.06	≈ 1080 N/mm²	**Tool steel** alloyed and pre-toughened, good cutting properties	Plates for mold tools and dies with increased requirements on strength

The inserts and slides can be made out of hot-work steel 1.2343.

1.2343	**DIN:** X 38 CrMoV 51 **AFNOR:** Z 38 CDV 5 **UNI:** X 37 CrMoV 51 KU **AISI:** H11	C - 0.38 Si - 1.00 Mn - 0.40 Cr - 5.30 Mo - 1.20 V - 0.40	≈ 780 N/mm²	**Hot-work steel** high-alloy	Molding plates and inserts for plastic injection mold tools
1.2343 ESU (ESR)	**DIN:** X 38 CrMoV 51 **AFNOR:** Z 38 CDV 5 **UNI:** X 37 CrMoV 51 KU **AISI:** H11 ESR	C - 0.38 Si - 1.00 Mn - 0.40 Cr - 5.30 Mo - 1.20 V - 0.40	≈ 780 N/mm²	**Hot-work steel** suitable for mirror polishing, electro-slag remelted, high-alloy	Molding plates and inserts for die casting (Al, Mg, Zn etc.) and plastic injection mold tools

If the quantities are in the millions, all of the plates and inserts are made of a through-hardening material, e. g. 1.2767 for the cavity plates.

1.2767	**DIN:** 45 NiCrMo 16 **AFNOR:** 45 NCD 16 **UNI:** 40 NiCrMoV 16 KU **AISI:** ≈ 6F7	C - 0.45 Si - 0.25 Mn - 0.40 Cr - 1.35 Mo - 0.25 Ni - 4.00	≈ 830 N/mm²	**Steel for through hardening** special alloy suitable for polishing, with high resistance to pressure and good flexural strength	High-performance cavity plates and inserts; cutting and bending inserts for high compressive loads

The inserts and the slides are also made of through-hardened steel 1.2379.

1.2379	**DIN:** X 155 CrVMo 121 **AFNOR:** Z 160 CDV 12 **UNI:** X 155 CrVMo 12 1 KU **AISI:** ≈ D2	C - 1.53 Si - 0.30 Mn - 0.35 Cr - 12.00 Mo - 0.80 V - 0.80	≈ 850 N/mm²	**Steel for through hardening** wear-resistant cold-work steel, high-alloy	Molding plates and inserts as well as wear plates or cutting dies with increased wear resistance

Surface

A further requirement is the surface of the finished plastic part. Not every type of steel is suited for a visible surface. For a high-gloss surface, tool steel should contain some chromium. Some suitable material grades are 1.2083, 1.2343 or 1.2344 ESR.

All three steels are also suitable for EDMed surfaces.

Surface

The quality of surface on the draft angle is also of high importance. No matter which machining method the surface was produced by, under a microscope they all look like a mountainous landscape.

Milling: Very often the milling direction is 90° offset to the demolding direction. This results in a stair-like milled profile over the entire surface and causes the formation of burrs resembling barbed hooks. The demolding becomes more difficult. Such milled surfaces must be re-treated. The most common post treatment is the polishing of the surface. In doing so it is polished in the demolding direction.

In Figure 3.34 the insert for our mold "container with cover" is milled.

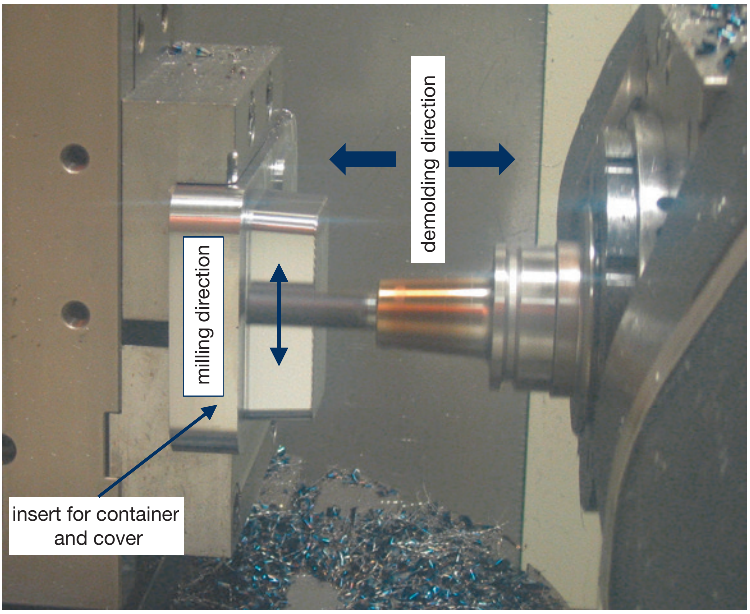


Figure 3.34 Milling insert [Factory picture: Gebr. Heller Maschinenfabrik GmbH, Nürtingen]

EDM: Surfaces produced by EDM—which is used, for example, to make ribs—are uniform but rough. Depending on the settings of the EDM machine the surface becomes more or less rough. The rougher the surface, the harder the demolding. Like the milled surface, an EDMed surface must also be re-treated. Polishing also helps here.

Wrong Side

If the orientation and the position of the component were incorrectly assessed, it is possible that the plastic part sticks to the fixed half. Or the already described vacpart falls out of the machine, the cover can be folded and directly pressed on the container.

Since the composed part would fill poorly through the 0.3 mm thick integral hinge, the container and the cover each injected with a separate nozzle. Both cavities fill and the flow line between the two melt fronts is in the integral hinge. The integral hinge will therefore break off. Different diameters in the nozzle can minimize the problem.

The cost for the use of a hot runner system with needle valve compared to an open system continues to increase. The mold is even bigger, and the overall cost for design, production and assembly is also higher here. And the system itself is also more expensive.

In the simulation shown in Figure 3.67 the flow line on the integral hinge can be seen.

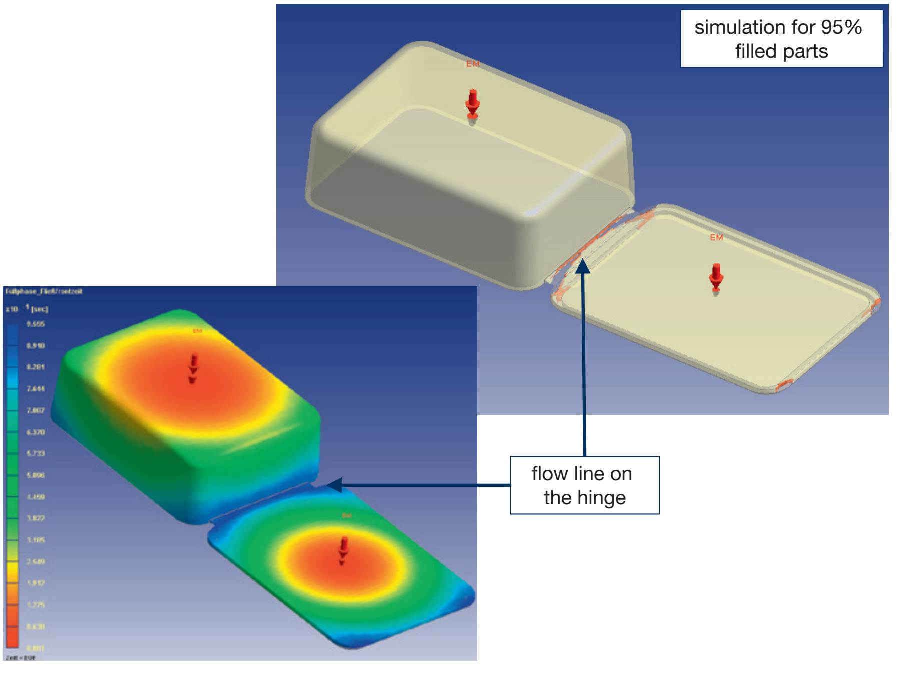


Figure 3.67 Simulation of uniform filling [Simulation: Cimatron GmbH, Ettlingen]

For a system with a needle valve, the filling can be controlled so that the flow line is moved either further inside the cover or further inside the container. Thus there is no flow line in the hinge and it will last longer.

In Figure 3.68 the different filling of both cavities can be seen. The cover fills up sooner than the container. The flow line is shifted further inside the container.

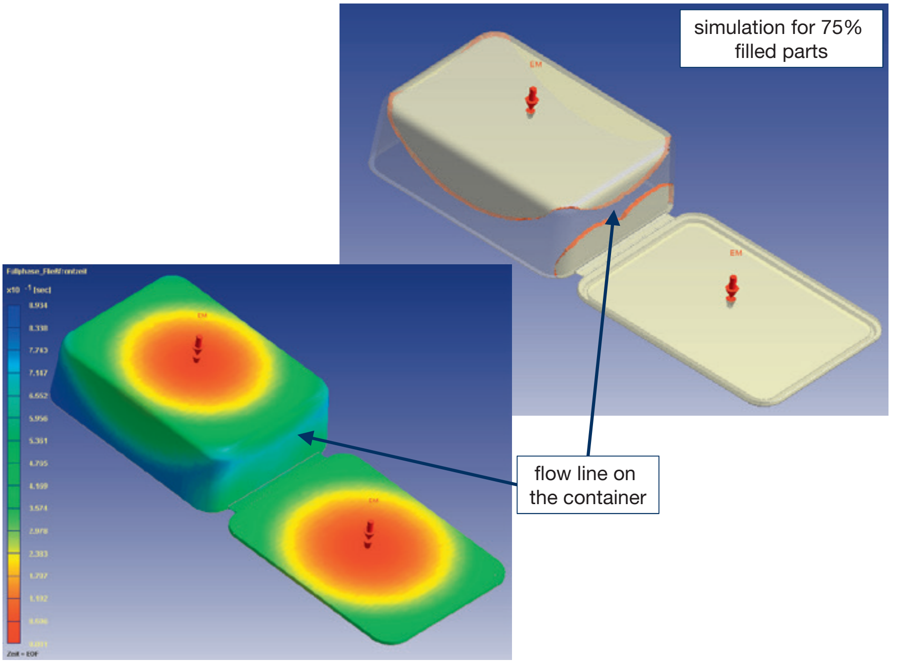


Figure 3.68 Filling with delay [Simulation: Cimatron GmbH, Ettlingen]

Another very important option results from the time delay control of the needles for very long and thin plastic parts. If these parts are injected uniformly with a hot runner distributor system with for example six open nozzles, five flow lines result. If the same part is filled with needle valve nozzles it could look like this: The first nozzle opens and fills the first portion of the cavity. If the melt front is over the second nozzle, it opens and injects into the flowing melt. This continues until the part is completely filled. The exact procedure must be determined through a filling analysis on the machine. Flow lines are avoided or minimized.

In Figure 3.69 the design of a hot runner system with needle valve nozzles is pictured.

In Figure 4.12 the hole in our slide, which is the undercut in our container which must later be demolded, is not connected with the split line face. Nevertheless the slide is placed below on the level of the main split line face. If you only want to demold the hole then the slide must go through the fixed half at the corresponding position on the side. The slide must demold during injection before the mold opens or otherwise the part is destroyed. This is technically feasible but the mold would be costly and unsafe for production.

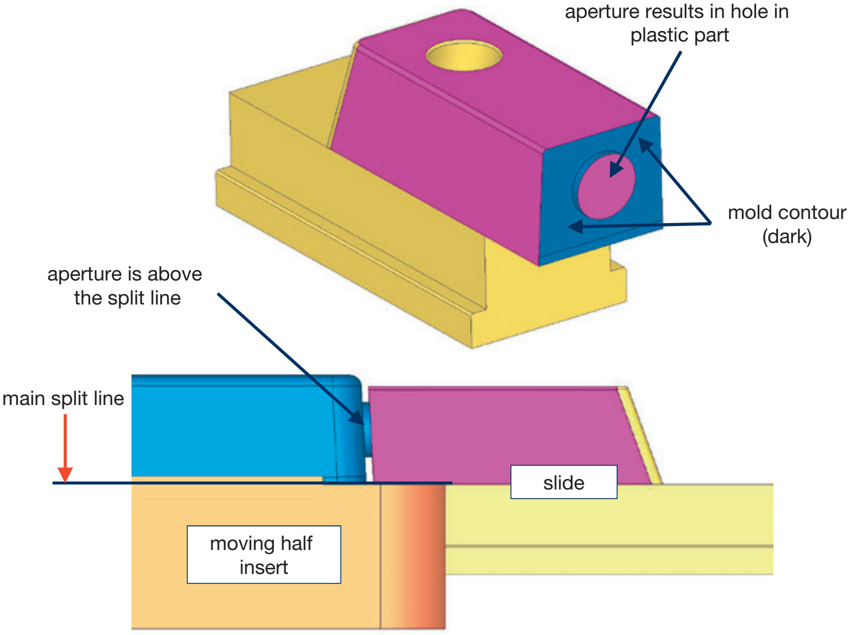


Figure 4.12 Undercut in the center of the plastic part

4.2.2.2 Split Line on Slide

If the mold contour is on the slide then the split line is circumferential to the mold contour. The split line must be tight but also serve as ventilation.

For the split line on the slide the same rules as for the general split lines in Section 3.8 apply.

There is an additional decisive factor for the slide. The slide or the split line of the slide moves forward and backward in the guiding. The problem there is the level split line at the bottom. With every movement the slide touches the level split line face. Because of the distance it must cover it can be that it moves beyond the edge of the moving half insert. During retraction in the mold the slide runs over the outer edge of the insert with the front edge of the split line. Over time this edge on the slide is damaged and a burr forms.

To avoid this, at this split line there should be a small ramp similar to the draft angle on the side of the slide.

> A split line on the slide should always be designed so that at the last moment of closing the injection mold it has contact with the opposite side. During opening it is reversed: it should be immediately free. Split lines that slip along the opposite side will not last very long.

In Figure 4.13 two slides from our container are displayed. On the big slide a circumferential split line is drawn. The inclined ramps are seen at the bottom of both slides.

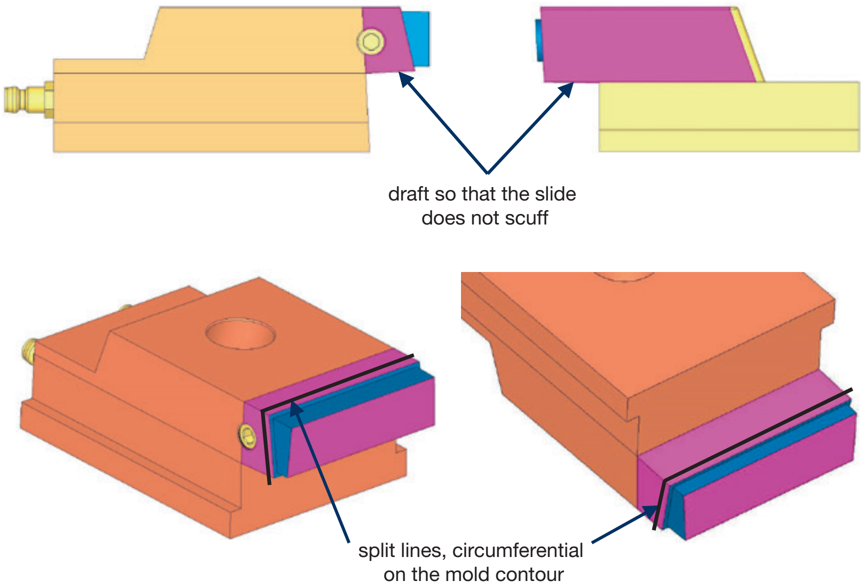


Figure 4.13 Slides with split lines and ramps

In Figure 4.14 the real situation is shown on the injection mold. The lead-in chamfer is shown on the slide and in the insert. The slide has first contact with the insert when the injection mold is totally closed.

The same applies for slides which are installed on the lower half of the mold split line. Such a slide, in addition to the inclined ramp below (see Figure 4.13), also needs a lateral ramp or lead-in chamfer. Through this lateral lead-in chamfer the split line between the slide and the insert is sealed. Because of the inclined split line, the slide has first contact with the insert just before the slide is in the first end position. On our container with cover from the integral hinge section there is

4.6.3 Graining

A treatment proven for decades to generate a structured surface on plastic parts is graining, or texturing, by etching. With this procedure a variety of surfaces can be produced, such as leather appearance, textile appearance, geometrical structures or also EDM structure.

For graining by etching, the inserts or cavity plates with mold contours are covered on all points which should not be textured with an acid resistant adhesive. The desired texture is inserted as a foil in the mold contour and covers the rest of the parts. The acid etches the clear areas. The textured surface is cleaned and sand-blasted. The foils are inserted again etching and sand blasting is repeated. The procedure is repeated several times until the final structure is achieved. The covered positions are freed again and the mold is finished. What is here relatively simply described is actually a complicated and elaborate process which demands time.

Since these structures are very deep the rules for the draft angle must be noted.

In Figure 4.64 a leathery texture and a grained EDM texture, which is generally known, are shown.

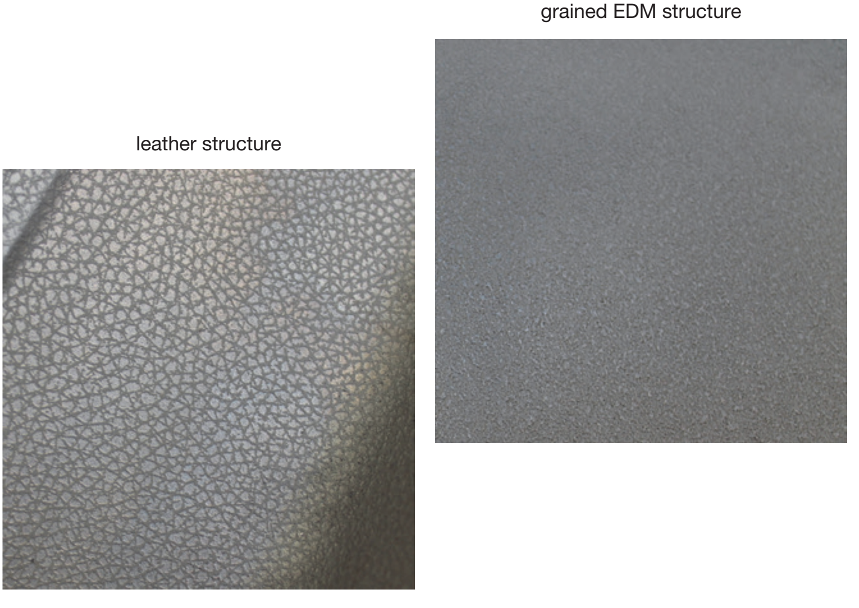


Figure 4.64 Different grained textures [Source: Reichle GmbH, Bissingen-Teck]

Example: Surface

Another example is the surface of the plastic part. As described in Section 4.6, the type of surface must be clarified with the customer at the beginning of the work.

Depending on the surface the preliminary work must be adjusted accordingly. If the surface is EDMed later, not too much effort and time are required in finish-milling.

If the surface is mirror polished subsequently, the EDMed surface is not a good working surface. Through EDM the microstructure on the surface is changed. During high gloss polishing this EDM scale must be completely removed or otherwise no shiny surface can be produced.

If in the planning for the construction of an injection mold the entire process chain is kept in mind and all steps are planned and communicated at the beginning of the project, then mistakes are minimized considerably. It is better to invest in precise planning at the beginning than to pay for costly rework or corrections later. In Figure 6.1 the process chain in mold making is graphically displayed. There are processes that depend directly on each other but there are also processes that are independent. In any case it is important to know the steps and how they are related to each other.

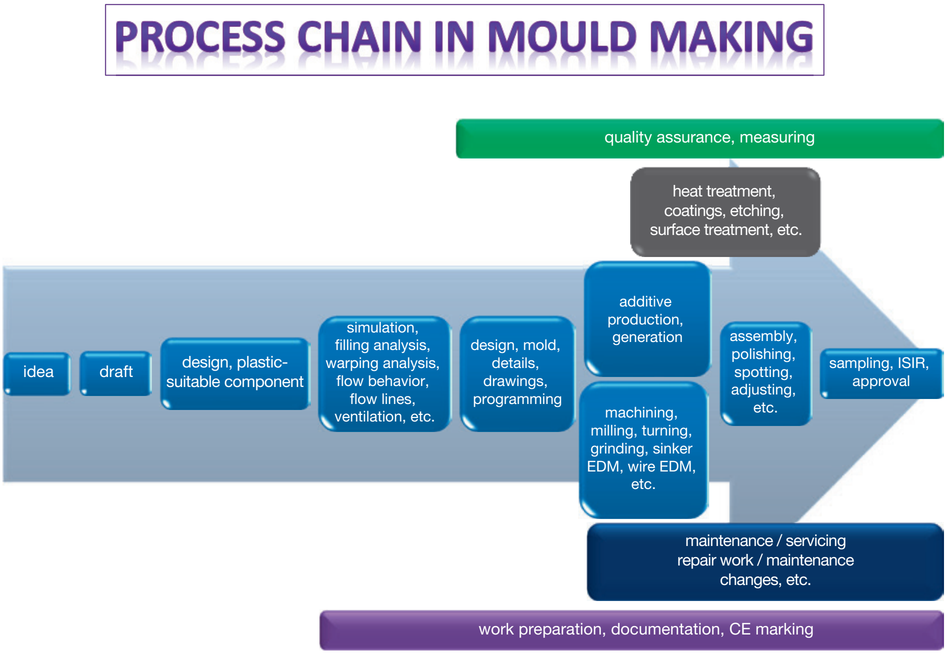


Figure 6.1 Process chain in mold making

■ 6.3 Quality Assurance

Quality assurance (QA) is more than, for example, checking the quality of an insert or measuring its width. Although this is also a part of quality assurance it is just a small part.

Mission Statement, Vision

Every company should define a mission statement or vision. These are goals the company wants to reach in an agreed time period, usually a calendar year. These goals are different depending on the company. For example, a 5% increase of turnover or the reduction of error rate to 0.5%, increasing the spindle hours in the milling department by 10%, acquiring new or specific customers, or being active in a new industry, etc.

Achieving these goals must not be left to chance.

Example: Reducing the Error Rate

Let’s look at the example of reducing the error rate. Errors always involve time and money. It’s not about looking for someone to blame. Possibly a process was defined falsely and could not be processed differently. If an error occurs, different mechanisms must be set in motion. What happened, why did it happen, and what can we do to assure it does not happen again?

- What happened? The ejector is too short.
- Why did it happen? The measurement carried out with the caliper was incorrect.
- What can we do to ensure that it does not happen again? Introduce an audit for measuring equipment or change the testing cycle of the measuring equipment.
- Of course the ejector must be remade but this is only a logical consequence.

What is described above must be an automatic process in which the hours, the value, and the costs incurred are determined. If the goal is the reduction of the error rate then the basis for the reduction must be known.

Description of Recurring Processes

In mold making, recurring processes and tasks are frequent. To maintain control of a company, it is necessary that the processes or tasks are described. This of course only applies for essential processes. It does not include, for example, how to find the way to the cabinet where the ejectors are kept. A sensible description of a process can for example say: each ejector has to be measured. At the beginning of the work, the measuring equipment must be checked with a gauge or caliper and the result recorded. The ejectors must be labelled; the measured dimensions must be off the list or written down.

The effort for following the process described is not much more than if this work would be excursively done. Even without process, a list of ejectors is needed, the measuring equipment must be available, and the ejectors must be labelled according to their length.

Assured Quality

If such processes are complied to then quality is ensured over a long period. Quality should never be left to chance.

A very important aspect of quality assurance is that every employee should follow the same process. Thus the quality is consistent and secured for the entire company.

The quality assurance is a live system. Its effectiveness must be constantly reviewed and checked. Insufficient processes must be expanded and too strict guidelines relaxed. Changes of the tasks, the environment, the techniques and so one must be adapted to the changing descriptions and conditions.

Consistent Quality Assurance

The list of the defining processes and procedures can be continued indefinitely at this point. Starting from the offer of the customer's ordering process, the process planning, the definition of the individual process steps from the procurement and the production to the dispatch of the product, it is important that the systematic quality assurance is understood.

Often the quality assurance is regarded negatively by a company as an additional cost factor. Of course this approach is wrong. By implementing consistent quality assurance and defining procedures and processes, mistakes are avoided, procedures are improved and simplified, and much becomes more transparent and clearer. The costs saved with the right quality assurance are higher than the costs incurred by it.

A company's success must never be left to chance!

Standards:

Different industries require different standards.

- The standard VDA 6.1 is applied in the German automotive industry.
- Suppliers who supply parts to the automotive industry, parts for driving vehicles must meet the standards of the ISO/TS 16949.
- The general standard which applies to mold making is the EN ISO 9001:2008.
- A guiding principle which must be valid everywhere is:

  Quality is when the customer comes back and not the injection mold.

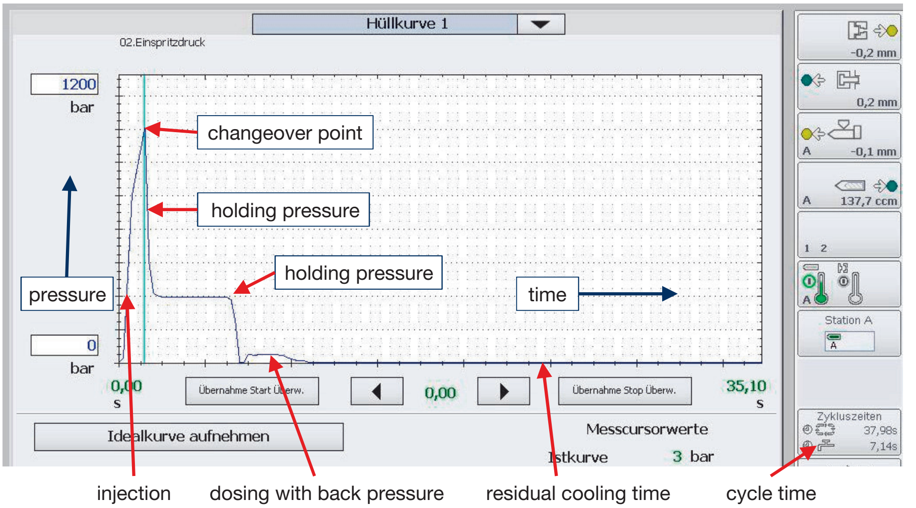


Figure 7.6 Complete cycle in process [Source: Friedrich Heibel GmbH Formplast, Heuchlingen]

7.1.4 Forces Acting in the Mold during the Injection Process

During an injection process, different, partly extreme pressures and forces act on an injection mold. Already during injection there is the injection pressure and then the holding pressure. For corresponding projected areas the cavity pressure can open a mold, that is, the closing force of the machine is not sufficient, or the mold becomes so deformed that it bends in the center when the split line is opened.

As already addressed, with high-viscosity plastic this can happen already during injection, that is, even before the holding pressure comes into effect at the changeover point.

A problem could also be an asymmetric distribution of plastic parts in a multi-cavity mold. If there are more or bigger parts on one half than on the other, there is an asymmetric cavity pressure through the projected area, which can lead to unilateral opening of the injection mold.

The same can happen if only one plastic part is in the mold, which has a thinner contour in the center and becomes bigger towards the outside.

The cavity pressure also affects the slides by lateral forces. As already described in Section 3.5, the cavity pressure works in all directions. In the demolding direction the machine supports the mold's stability. However in the direction of the slides the mold itself must absorb the forces. Correspondingly dimensioned plates and cotters function as counterholders.

The above described for the slides also applies for inclined or jumping split line faces. Here there are also lateral forces through the cavity pressure. Counterholders or additional locks can provide support here.

An additional problem affecting the cavity pressure acting in all directions can be the cooling and the ejectors. When the cooling channels or the holes for the ejectors are too close to a wall, the constant loading and unloading during the injection process can cause a crack in the thin wall between the cooling and the component wall. The consequences: either the cooling circuit leaks, or in the case of the ejector holes the ejector moves with difficulty with over time and thereby gets jammed. The temperature fluctuations through the hot melt aggravate this problem.

The following Figure 7.7 is taken from Section 3.3 to better illustrate the forces generated by the cavity pressure.

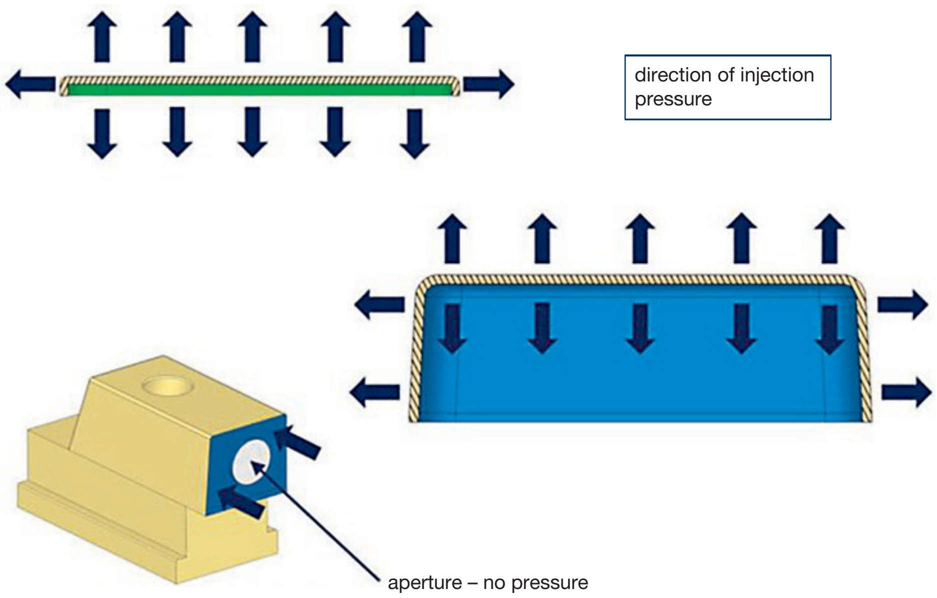


Figure 7.7 Forces and direction of forces generated by cavity pressure

7.1.5 Initial Sample Inspection Report

The machine is set up and the first components are finished. Now the quality and the assured characteristics of the plastic part must be documented for and confirmed to the customer.

The initial sampling is for this purpose. As the name suggests, this is the initial test. All further tests are follow-up tests. For the initial sampling, all the dimen-

Compared with the finished (prefabricated) part, a generated core has still some allowance on it so the finished contour can be produced.

The advantages of laser sintering are:

- Time saving: Only a few hours after the design, the core can actually be held in hand, no matter how complex it is.
- Flexibility: Modifications of the design data and production of variants can be integrated in the original process and quickly implemented after a sampling.
- Quality: With the laser more complex molds can be generated than with traditional methods. This can improve the quality of the products.
- Productivity: The installation of conformal cooling systems in injection molds enables the reduction of the production cycle times.
- Cost reduction: As a general rule, laser sintered inserts from original material last longer than conventionally produced inserts. In connection with the aforementioned factors this results in clear cost benefits.

Figure 9.22 displays a core on the laser machine on the left, with some powdery material on the right of the core. The right shows the finished, mirror polished mold core and the produced plastic parts.

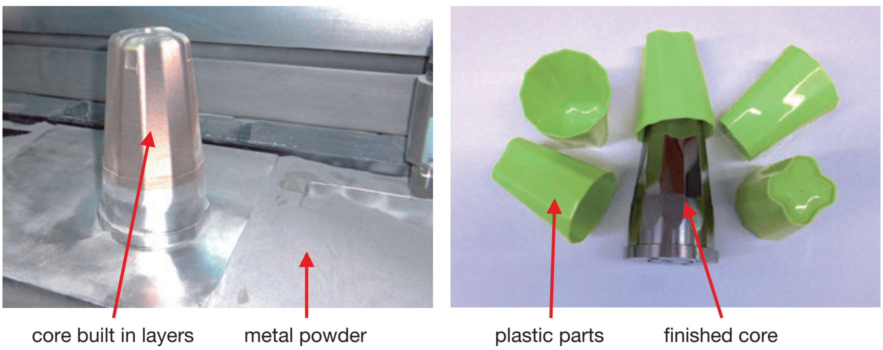


Figure 9.22 Raw material and finished core [Source: bkl-lasertechnik, Rödental]

9.6.2 Vacuum Soldering

Like laser sintering, vacuum soldering belongs to the recent technologies in mold making. The soldering itself is a long-established practice for connecting materials. The trend towards conformal cooling was a motivation for the development of this method.

During vacuum soldering several individual parts are soldered together with a solder. In practice it can look like this: many layers of steel are laid one above the

Index

Symbols

A

B

C