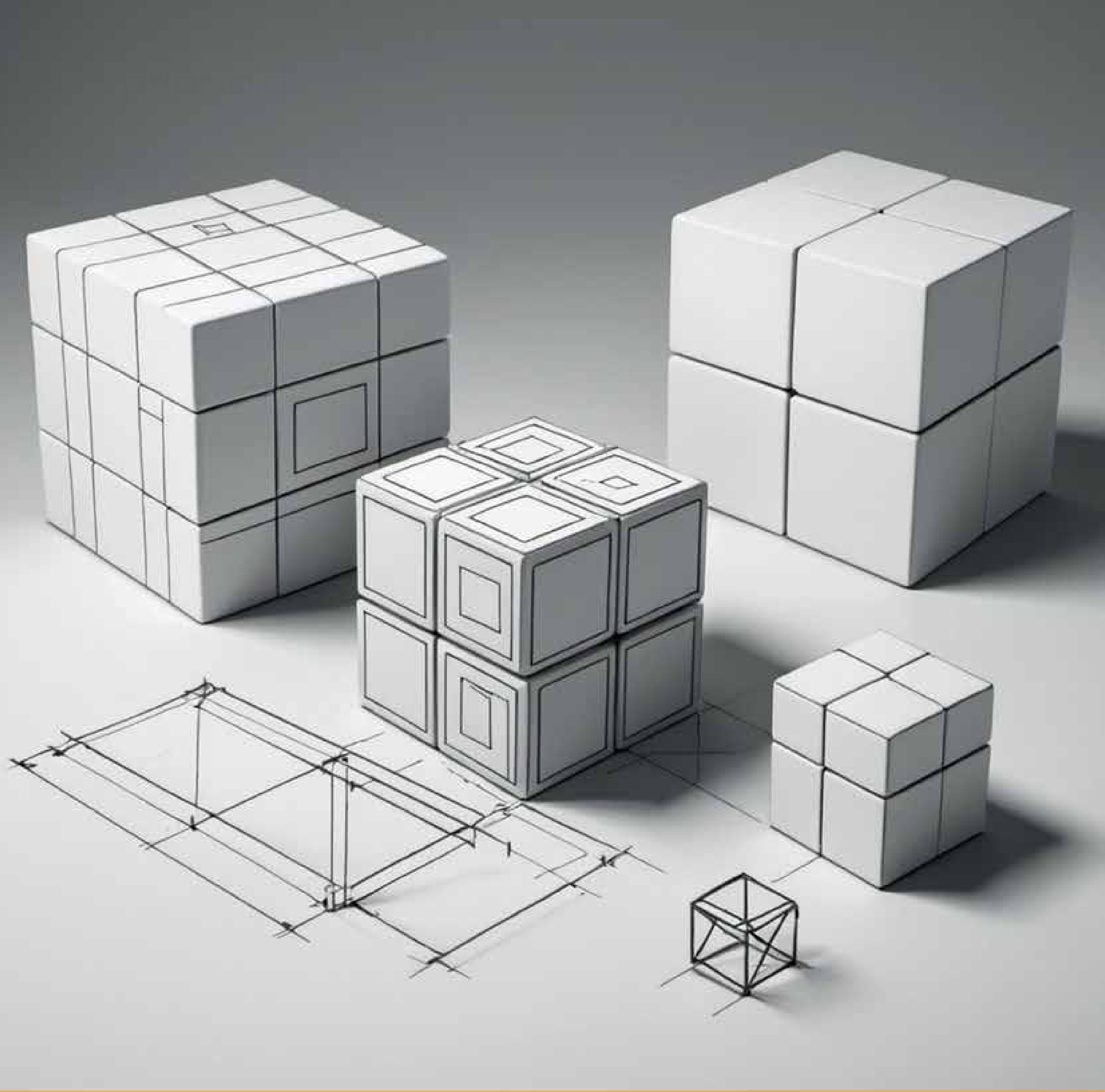


DESIGN GUIDE

HOW TO CREATE THE PERFECT PROTOTYPE

A step-by-step guide to take
your injection moulded prototype to the next level

priomold
KUNSTSTOFFSPRITZGUSS

Contents

Introduction - Highest quality in the shortest time

Plastic parts have become essential in today‘s world. They are used in household items, technical components for the electrical and automotive industries, and products for ophthalmic optics or medical technology, to name but a few. This widely used material allows components to be moulded and gives consumer goods a wide range of properties, from flexibility and pliability to strength and impact resistance.

However, producing a plastic part can be a time-consuming process. First of all, you need to have a thorough understanding of the product design. After that, you need a manufacturer who can produce the part. The search is often tedious, especially for prototypes or low-volume production, as many injection moulders are not interested in producing „only“ 500-1000 parts.

As a result, many switch to additive manufacturing processes and then realise that the parts do not have a satisfactory surface without extensive post-processing, do not meet the necessary strength requirements or the number of parts required exceeds the cost of the project.

This is where priomold‘s expertise becomes clear: close-to-production prototypes made of plastic as well as complete pre-series and small series are produced in a short time - on average, two to four weeks are required to complete a new mould.

Advantages of the rapid tooling process:

- Rapid tooling enables cost effective mould modifications. Surface and material quality is at the same level as in series production.
- The process covers a wide range of applications, from simple components to more complex structures such as automatic slides or insert and two-component injection moulding.

The rapid tooling process is used to produce injection moulds from high-strength aluminium using HSC milling machines. Mould construction is similar to conventional toolmaking, but priomold is characterised by a special master mould system, which means that many tool components can be saved. A wide range of thermoplastics, including ABS, PC, PA66, PA6, PC-ABS, POM, PP, PE, PEEK, with various fillers such as glass fibres, glass beads, carbon fibres, mineral fibres and other variants, can be processed on priomold‘s injection moulding machines. Thermoplastic elastomers that are particularly flexible, such as TPE or TPV, are also processed.

This guide provides an insight into the injection moulding production process and highlights the differences from conventional mould making. It serves to adapt the product requirements to the rapid tooling process. Furthermore, key design aspects are explained in order to avoid common errors in injection moulding and thus improve the quality of the manufactured parts. These measures can avoid time-consuming feedback loops in the project phase and reduce unexpected costs for mould changes.

In addition, quality management and post-processing options at priomold will be presented. Here, reference is made to an internal measuring laboratory and a regional network for post-processing such as painting and coating, which enables comprehensive support from a single source.

The priomold design guide provides insights into the injection moulding production process as well as important design aspects to avoid common mistakes. In addition, the experienced priomold team supports the entire development process through to optimisation for series production.

Injection moulding - the basics

The first section presents an in-depth introduction to the injection moulding process and a comprehensive analysis of the mould structure. When designing injection moulded parts, the focus is on the pre-optimised consideration of the following aspects:

- Design of the injection points
- Positioning of the ejector pins - which can also be used as venting points

The strategic optimisation of the injection mould design and the production process leads to a win-win situation by increasing productivity and improving the cost-effectiveness of plastic part production.

The injection moulding process and how it works

Injection moulding is a mould-based production technology that has proven to be extremely efficient and precise. To produce an injection moulded part, the process begins with the feeding of plastic granulate into the hopper. Through the combination of heat and pressure, the granulate is melted. Heat is provided by heating bands on the barrel, while the screw with variable pitch generates the necessary pressure.

The screw conveys the melted plastic from the end of the barrel towards the mould. Once the molten pellets are viscous enough, the screw moves forward and the molten material is injected into the mould. The moulded part cools and takes on the exact geometry of the mould tool.

Once the cooling phase is complete, the mould is opened and the finished injection moulded part is ejected via the ejector unit.

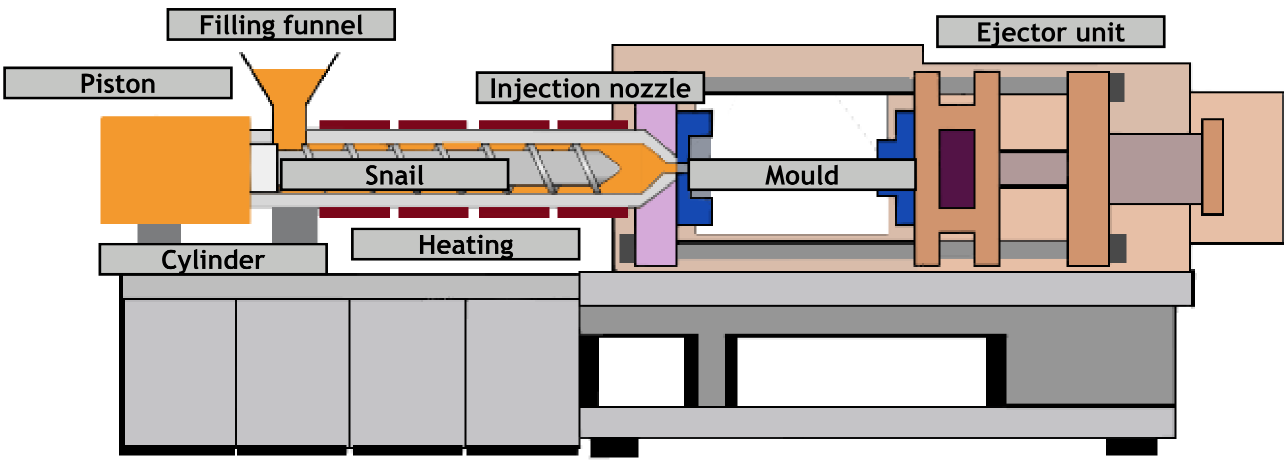


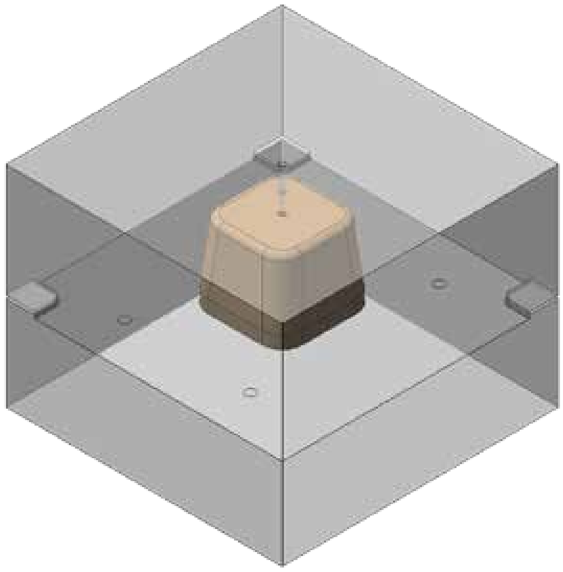
Illustration of the structure and function of a plastic injection moulding machine

Tool and mould design

The two sides of the mould

The mould of an injection moulded part consists of two main sides: the A-side and the B-side. The A-side, also known as the nozzle side or visible side, fulfils higher visual requirements and can be polished, blasted or finely milled according to design specifications.

In contrast, the B-side or ejector side is the functional side and contains the hidden structural elements of the part (ribs, snap-fit connections, etc.). This often has a finely milled surface and visible traces of the ejector.

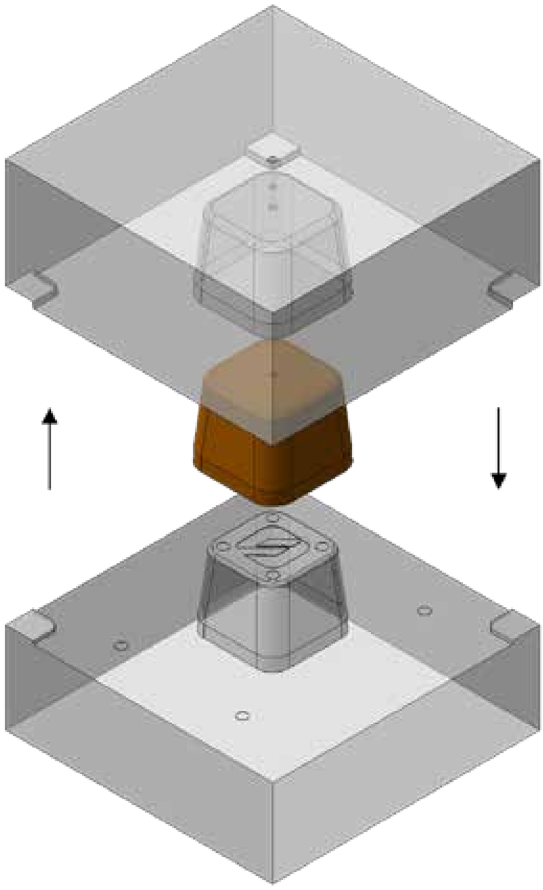


Closed mould in the injection moulding process:

View of the A-side and B-side

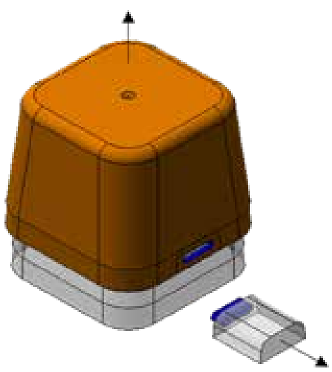
The sliders

For part designs with undercuts that cannot be demolded in the standard open-close direction, sliders are used. These components move laterally within the mold to release the undercuts before the part is ejected.Sliders should be used sparingly as they increase the complexity and therefore the overall cost of the mould. An important tip is that the slider should move parallel to the parting line and draft angles must be added as normal.



Open mould in the injection moulding process:

View of the A and B sides and ejector

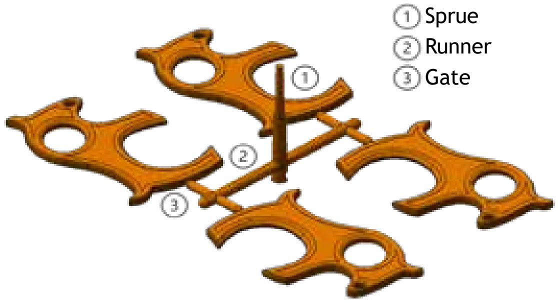


How the sliders work

The gating system

The runner system is the path through which the molten plastic enters the mould. It normally consists of a sprue (main channel), runners (guide channels) and gate (entry points). After ejection, the sprue system is separated from the component.

Different gates are suitable for different applications:



Gating system of a 4-cavity injection mould

Spot gating with three-platen moulds:

Spot gating in three-platen moulds injects from the visible side of the part. Ideally suited for the injection of lids or housing parts. An inexpensive alternative to the hot runner for small quantities.

Bar sprue moulding:

Bar sprue is a simple type of sprue with a round cross-section that merges into the cavity at the thickest point of the workpiece. Suitable for components with a large volume. After casting, this area must be machined and remains recognisable despite machining, which is why it is not applied to visible surfaces.

Tunnel sprue:

The tunnel sprue is a self-separating sprue system in which the moulded part is injected from the side and separated from the sprue during ejection. The sprue is located in the nozzle-side mould half and the runners end in a conical bore. When the mould is opened, the injection part and sprue system remain on the moving mould half and a sharp cutting edge separates the tunnel sprue from the injection moulded part. Demoulding is then carried out by the ejector.

Tape or film sprue:

Tape or film gating is used for large-area or very thin parts, as it leaves no gate marks and provides parallel molecule orientation. This ensures uniform shrinkage in the flow and transverse direction and uniform filling of the mould cavity. After casting, the sprue must be removed.

The gate

At the point where the runner system is connected to the part, a small imperfection is usually visible – this is referred to as the gate. Mark areas where this is less disruptive or try to integrate the bleed into your design.

If the presence of the gate is not desirable for aesthetic reasons, it can be „hidden“ in the functional B-side of the part, depending on the part design.

The clamping and ejector unit

The clamping and ejector unit on the opposite side of the injection moulding machine has two functions. It holds the two moulded parts together during injection and ejects the component after opening. However, the alignment of the various moving parts of the mould is never perfect. This leads to the following visual defects, which are visible on every injection moulded part:

Ejector marks that are visible on the hidden B-side. They occur because the ejector pins protrude slightly above the surface of the mould.

Separation lines that are visible on the side of a part where the two halves of the mould and sliding elements meet. They are caused by tiny misalignments.

TIP:

- Mark important sliding or sealing surfaces in your drawing
- Specify the parting plane and place it at defined edges to avoid parting lines

Typical optical defects of an injection moulded part

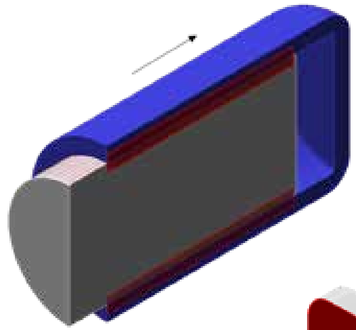
Aluminium mould in plastic injection moulding

Take your design to the next level

The following guidelines will help you to improve the quality of your prototypes and reduce possible failures in production.

The most common injection moulding defects

Draw marks

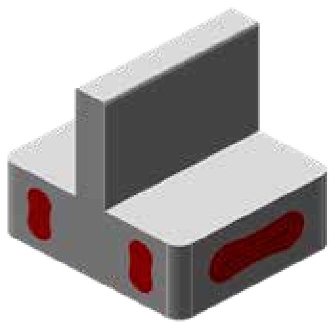


When the plastic shrinks, it applies pressure to the mould. When the mould is ejected or opened, the walls of the part slide and scrape over the mould. This leads to abrasion on the injection moulded part. Parts without a demoulding angle and vertical walls are most susceptible to this defect.

Warpage

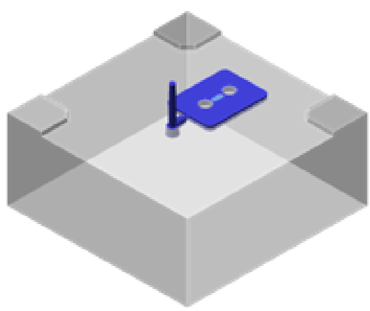
If certain sections cool down faster than others, the part may warp permanently due to internal stresses. Injection moulded parts with different wall thicknesses have the greatest tendency to warp.

Sink marks



If the inside of a part cools down and no more plastic can flow in before the mould is completely filled, small indentations called sink marks occur. Thick walls or poorly designed ribs are the most common reasons for sink marks.

Weld lines

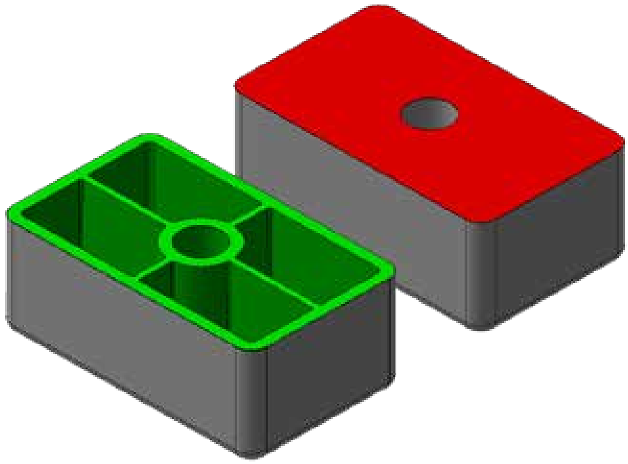


When two material flow fronts meet, small line-like discolourations can occur. The weld lines have a cosmetic effect, but generally reduce the strength of the part. Holes or abrupt changes in geometry are the main reason for weld lines.

Best practice

Constant wall thickness

Always design parts with a constant wall thickness that is as thin as possible to avoid sink marks and warping. As a guideline, we recommend a thickness of 1 mm to 3 mm. Please note that every 10 % increase in wall thickness leads to an increase in rigidity of ca. 30 %; if thicker cross-sections are required for functional reasons, these should be cut out and stiffened with ribs instead.

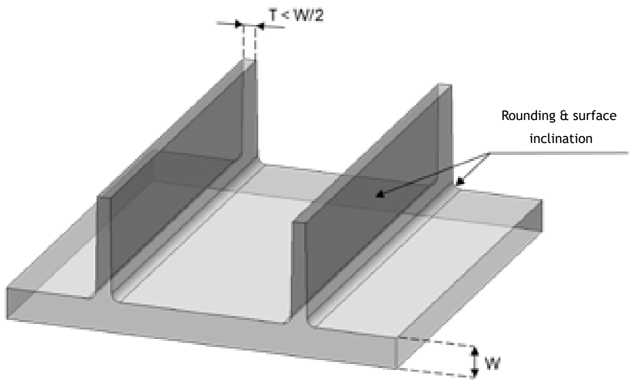


Ribs

If the maximum recommended wall thickness is not sufficient to fulfil the functional requirements of your component, ribs can be used to increase its rigidity.

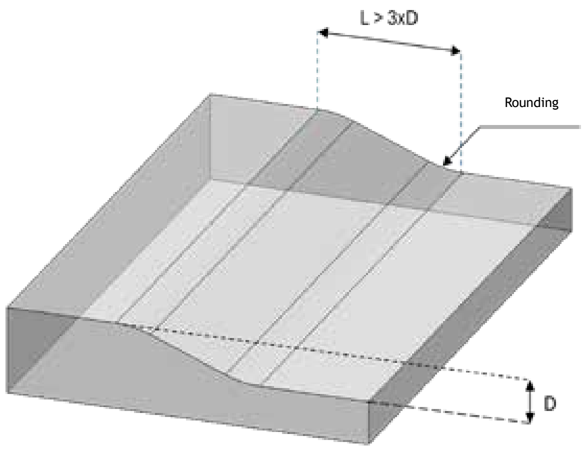
Pay attention to the following points when designing to avoid errors in your part:

- Use a thickness of about 50% depending on the main wall thickness
- Add an angle of inclination of at least 0.25-0.5°
- Use a basic rounding with a radius greater than ¼ × rib thickness



Smooth transitions

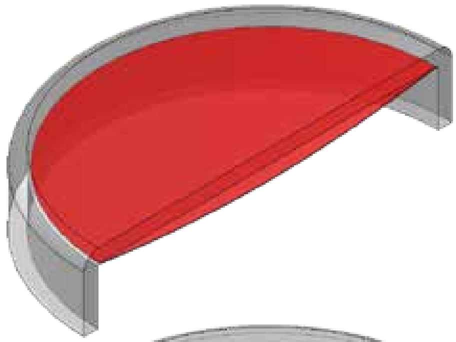
In some situations, cross-sections with different wall thicknesses can hardly be avoided. In these cases, you can smooth the transitions with roundings or chamfers to counteract distortion caused by different shrinkage. We recommend three times the wall thickness difference as a guideline.

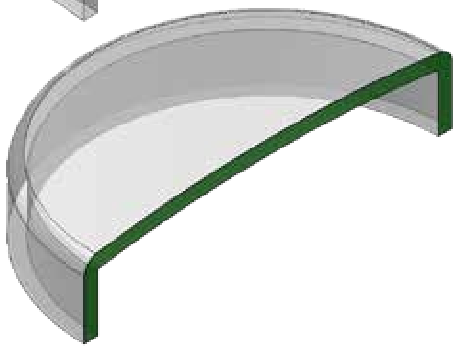


Avoid flat surfaces

Flat surfaces, especially flat visible surfaces, should be avoided as far as possible, as they tend to collapse (0.5 mm to 1 mm per 100 mm, depending on the material).

Due to uneven shrinkage, smooth, flat surfaces usually appear uneven. Design curved surfaces. Although this does not prevent shrinkage, it only has the effect of changing the curvature and is therefore usually no longer noticeable. It is also possible to apply ribs to counteract warping. However, there is a risk of sink marks on the visible side due to mass accumulation at the base of the ribs.

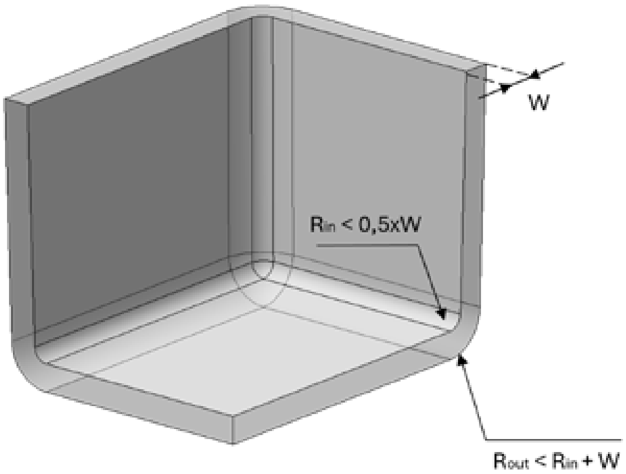




Rounding edges

Apply the rule of constant wall thickness to the corners of the part as well. Add as large a radius as possible to all outer and inner edges.

Due to their low flow resistance, curves facilitate the flow of the melt and favour uniform mould filling. This reduces the tendency for flow lines. Another advantage is better demoulding due to the reduced notch effect.



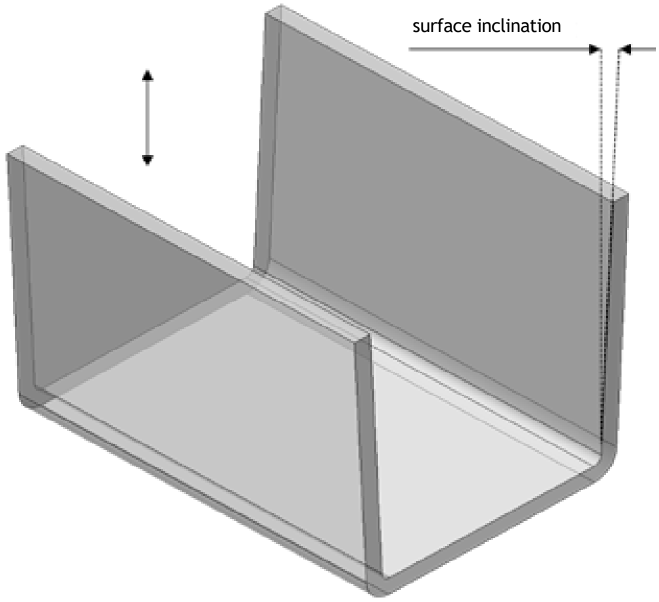
TIP:

- Add a rounding to the outer edges that corresponds to 1.5 times the wall thickness
- Add a rounding to the inner edges that corresponds to 0.5 times the wall thickness

Add demoulding angle

Add a demoulding angle to all vertical walls to make it easier to eject the part and avoid pull marks when opening the mould.

Guide values for the average draft angles of different plastics are as follows:



Amorphous

Plastic type	Demoulding slope (°)
ABS	0,5
PC	1-1,5
PS	1-1,5
SAN	1-1,5

Increase the demoulding angle for structured parts to at least 2 to 5° depending on the surface quality.

Semi-crystalline

Plastic type	Demoulding slope (°)
PA 6.6 GF	0,25-1
POM	0,25-0,5
PE-HD	0,5
PP	0,5
PET, PBT	1

Surface quality

The surface of the injection moulded parts is generally not reworked, but the mould itself can be finished in different ways. In this way, aesthetic requirements (e.g. a matt surface) or technical requirements can be fulfilled. We offer you the following finishing options: Post-processing by blasting surface finish similar to VDI 3400.

Designation priomold	Similar VDI 3400	Required demoulding slope (°)
S330	Ref.33	5
S280	Ref.30	4
S170	Ref.28	3,5
S110	Ref.27	3,5
S70	Ref.24	2,5
Glass beads	Ref.20	2

The required mould bevel should be seen as a minimum value in order to avoid drawing marks on the component!

Other surfaces

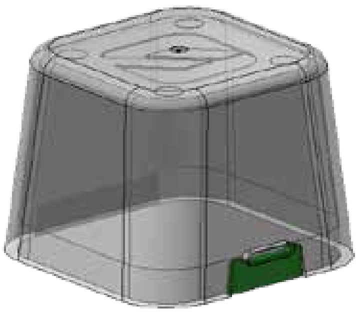
	Description	Application
Finely milled	The priomold standard. Mould marks may be visible.	For functional parts without visual requirements. Standard for processing the B-side.
Polished	The mould is roughly polished in the demoulding direction, improving the demoulding of the component.	For components with a small demoulding angle. Usually used on the B-side.
Technically polished	The mould is finished with fine-grained sandpaper, giving you parts with a fine surface finish.	Suitable for parts that require a good visual appearance but not a high gloss.
High-gloss polished	The mould is smoothed and then treated with polishing paste, giving you parts with a mirror-like surface.	Suitable for parts that have very high cosmetic requirements or for parts in the field of optics (e.g. fibre optics).

Avoid unnecessary undercuts

Undercuts are features that cannot be produced with a simple two-part mould, as material would be in the way when the mould is opened or ejected.

Unnecessary undercuts should be avoided as they usually require a great deal of mould technology and result in high mould costs.

The following examples show how this is done. Basically, material is removed from under the undercut so that it can be moulded in the open-close direction.



It's all a question of separation

The easiest way to deal with undercuts is to move the parting line of the mould so that it intersects them.

This option is suitable for many designs with undercuts on an outer surface. Make sure to adjust the demoulding angles accordingly.

Below you will find simple solutions for dealing with undercuts:

Use breakthroughs

Another way to deal with undercuts is to remove material above or below the problem area. This removes the undercut as the contours are mapped directly from the A and B sides.

Cutouts are perfect for dealing with undercuts on the inside or sides of the part.

Consider tolerances

When injection moulding priomold, deviations in accordance with DIN 16742-TG5 (plastic mould part tolerances as symmetrical limit dimensions for size dimensions) are to be expected. You will find the corresponding table below:

	Limit dimensions in mm for nominal ranges in mm												
	1 to 3	> 3 to 6	> 6 to 10	> 10 to 18	> 18 to 30	> 30 to 50	> 50 to 80	> 80 to 120	> 120 to 180	> 180 to 250	> 250 to 315	> 350 to 400	> 400 to 500
W	±0,05	±0,08	±0,11	±0,14	±0,17	±0,20	±0,23	±0,36	±0,50	±0,58	±0,65	±0,70	±1,00
NW	±0,08	±0,11	±0,14	±0,17	±0,20	±0,23	±0,36	±0,50	±0,58	±0,65	±0,70	±1,00	±1,40

Tighter tolerances are possible under certain circumstances. These can be kept in the mould by arrangement and modified after the initial sampling as part of a chargeable tool modification.

TIP:

- **Tolerance your component only as precisely as necessary.**
- **Always design your components to the centre of the tolerance in order to be able to comply with the limit dimensions and avoid correction loops.**
- **Use levelling elements such as pinch ribs instead of fits.**

Fonts

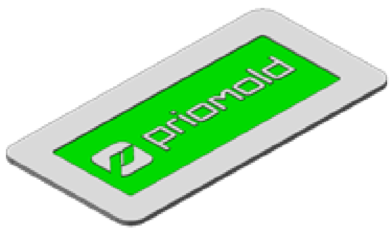
Raised lettering: Raised lettering is the easiest to produce, as it only needs to be engraved into the mould. However, raised lettering can interfere with a flat surface. (Support or mounting surfaces)

Recessed raised lettering: When using a stamp with engraved lettering, the raised lettering can be set deeper in the moulded part and therefore does not interfere.

Recessed lettering: Recessed lettering is the most complex to produce, as the characters have to be milled out in relief in the mould insert.

This variant should only be used if raised or recessed raised lettering is not acceptable for design reasons.

Raised lettering



Recessed raised lettering

Recessed lettering

TIP:

- Ensure that there is sufficient space between the signs to avoid gaps.
- We recommend a minimum distance of 0.3 mm.

Optimisation, finishing and quality assurance in injection moulding prototyping

Optimisation and further development

From product development to series production, the priomold team offers expert know-how right from the start of the project and accompanies the development process through to the optimisation of a plastic part suitable for injection moulding for series production.

Support is provided right from the start with the selection of materials. Help is offered with the optimisation of injection moulding-compatible designs for plastic parts and even the design of injection moulded parts based on drawings or sample parts if it is not possible to create corresponding models.

Finishing and shipping

We use documented test instructions and internal retention samples for both initial sampling and ongoing production to ensure that the parts are always of consistent quality. Our specialists inspect the injection moulded parts in-house and pack them in accordance with the customer‘s packaging instructions.

In addition, priomold has an extensive regional partner network for post-processing services for plastic parts, such as painting, coating, printing and the procurement of insert and outsert parts for your products. This eliminates the need for many arrangements with different suppliers and offers an integrated product range from a single source.

3D scan and quality assurance

The fast throughput times for prototypes and small batches should not end with the completion of injection moulding production, but should also continue in downstream quality assurance processes. For this reason, we have had our own measuring laboratory with a GOM 3D scanner, manual measuring equipment and state-of-the-art software for creating initial sample test reports for our customers since 2020.

With our GOM Atos Capsule Scanner, we can digitise the smallest filigree plastic parts through to housing components using various measuring fields. Compared to conventional tactile measuring methods, deformations in the component can be detected much better and it is possible to carry out measurements on the digital twin of the injection moulded part at any time.

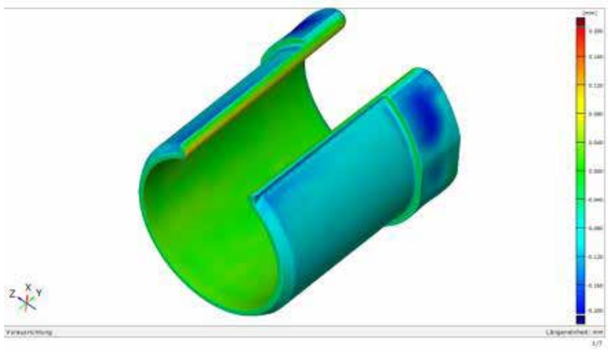
It is also possible to design a plastic-compatible model based on the scan data of an existing sample part.

Quality assurance with the GOM Atos Capsule Scanner

Evaluation

False colour target/actual comparison

For the qualitative evaluation of your plastic parts, STL mesh data is created from the 3D scan data and superimposed on the target data in a false colour representation. This representation can be used to measure specific measured values or deviations as well as to display distortion, flatness or position tolerances of injection moulded parts. For measurements of injection moulded parts during series production, the evaluation programs can be saved and made available to our customers for measurement or archiving.



False colour target/actual comparison

Initial sample test reports

Traditionally, so-called initial sample test reports with a measurement report are created for injection moulded parts for approval. As part of the initial sampling of the parts, various sub-areas such as geometry and dimensional testing or a functional test can be carried out. The main part of the initial sample test report for plastic parts comprises the dimensional inspection in accordance with the required drawing dimensions or our customers' specifications.

In addition to the services mentioned above, we also offer the creation of IMDS entries and support you in the creation and dimensioning of plastic parts. Effective dimensions for plastic injection moulded parts are very different from those of classic drawing parts (milling, turning, etc.).

Conclusion

In summary, it can be said that injection moulding-compatible component design depends on a variety of factors and must be individually adapted to the respective component. Successful plastic injection moulding prototyping requires a complete approach, starting with the selection of tool technology, through post-processing and quality control, to the constant optimisation and further development of the prototype design.

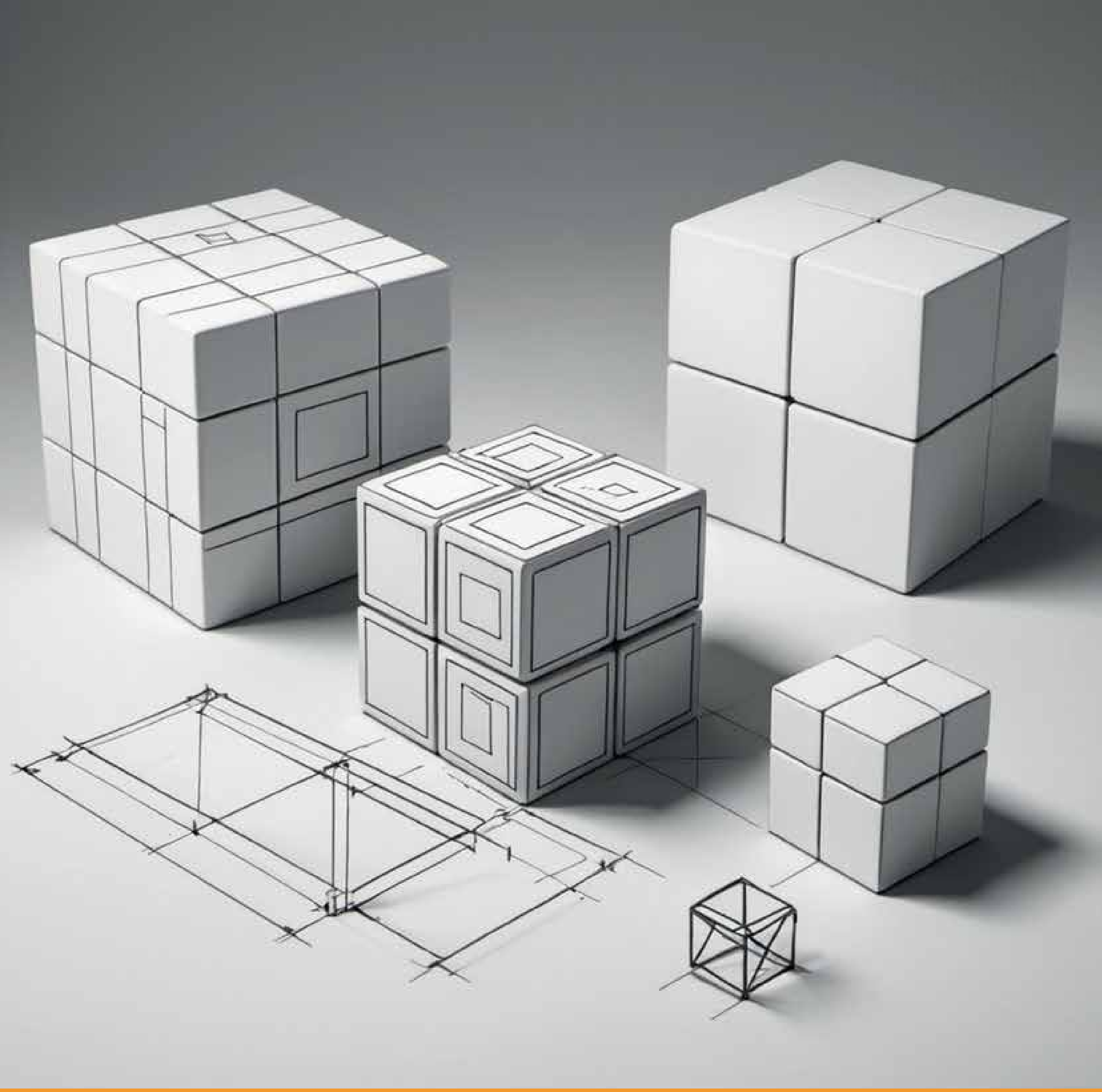
The main goal is to keep tool costs as low as possible and avoid costly modification grinding, while at the same time achieving high component quality. The trade-off between quality and cost is a considerable challenge. Involving an experienced partner in the design and manufacture of injection moulded parts in the early project phase can make a significant contribution to the optimal development and production of your component. A holistic approach ensures the production of high-quality injection moulded parts that both meet quality standards and fulfil the requirements of the project.

About priomold GmbH

The still young company, which was founded by Thomas Schönbucher and Moritz Zumdick in 2015, specialises in the rapid delivery of plastic injection moulded parts, offers toolmaking (over 500 new tools per year) for prototypes and small series as well as engineering support in the plastics sector. The company has now grown to over 80 employees and continues to expand. priomold sets itself apart with its short delivery times for tools, injection moulded parts and additively manufactured components. The fastest project was realised in two working days; on average, a new mould is completed within two to four weeks.

About the author

Stefan Geist, an experienced project manager and design engineer, has been with priomold GmbH for 7 years. His professional career began with an education as an industrial mechanic, followed by a technician qualification at the Heinrich Wieland School in Pforzheim. Since joining priomold, he has established himself as a key member of the company. Stefan is characterised by his comprehensive expertise in design and project management. His skills and experience contribute significantly to the successful realisation of complex projects. With a solid background as an industrial mechanic, Stefan Geist brings a practical perspective to his design projects. His technical training and many years of experience at priomold make him a reliable and experienced expert in the industry.



priomold GmbH
Gewerbestr. 6
75328 Schömberg

+49 7084 97 6969 0
info@priomold.de
www.priomold.de

priomold
KUNSTSTOFFSPRITZGUSS