N&H Technology GmbH

PLASTIC - MOLDED PARTS

System supplier for HMI Operating units

ENGINEERING | MANUFATURING | DELIVERY

COMPLETE SOLUTIONS & ASSEMBLIES

KEYBOARDS, KEYPADS & BUTTONS

CABLE ASSEMBLY & CONNECTORS

MOLDED PARTS & TOOLMAKING

TÜVRheinland
ZERTIFIZIERT
Management
System
ISO 9001:2015
www.tuv.com
ID 9105016435

N&H Technology GmbH

2001

Foundation of the N&H Technology GmbH with 4 employees in Krefeld

2012

New construction of a company building in Willich with its own test laboratory & logistics warehouse

2013

Opening of the N&H office in Shanghai

2021

Warehouse extension with 470 additional pallet spaces

2023

Majority interest SNT Technology Co.,Ltd. In-house design & production of input solutions

2025

Our N&H team comprises 51 employees - 13 of whom are *engineers**.

- Your expert for input units since 2001
- Over 5,000 realized customer projects
- Long-standing partnerships with leading companies from a wide range of industries
- In-house development & exclusive production for maximum flexibility
- Certified processes for highest quality

WE AT N&H TECHNOLOGY

Since our foundation in 2001, we have established ourselves as a leading full-service provider of customized electromechanical assemblies, moulded parts and components for individual operating units (HMI).

We offer comprehensive manufacturing solutions through our established supplier network in Asia, which is coordinated on a project-specific basis in cooperation with our subsidiary in Shanghai.

Our partners meet industry-specific standards such as DIN ISO 9001, ISO 14001, IATF 16949 and DIN ISO 13485, and our own test and inspection laboratory at our site in Willich complements our strict quality controls.

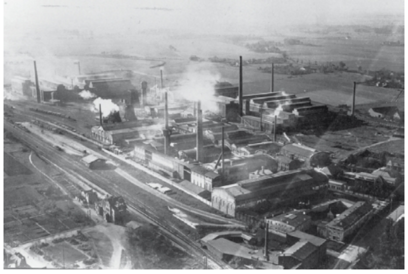
In 2023, we expanded our expertise in membrane switches by acquiring a majority stake in the highly specialized FoShan SNT Electronics Technology Co, Ltd. in China. This enables us to respond even more specifically to the individual requirements of our customers and offer high-quality solutions.

Our customer base includes leading companies from the automotive industry, medical technology, telecommunications, industrial automation, building control technology and other sectors. We are characterized by long-standing partnerships and a high level of customer satisfaction.

Our employees are the heart of N&H Technology and the key to our success. We support our international, family-like team and create an environment that encourages personal development, innovation and collaboration.

N&H Technology stands for innovation, quality and reliability - your
trusted partner for electromechanical solutions.

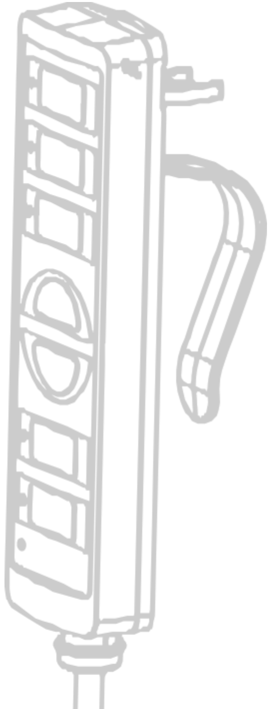
TÜVRheinland
ZERTIFIZIERT
Management System
ISO 9001:2015
www.tuv.com
ID 9105016435



Modernity meets history

Since the turn of the millennium, the site of the former Becker steelworks has been transformed into a diverse industrial park, where carefully restored monuments alternate with modern architecture.

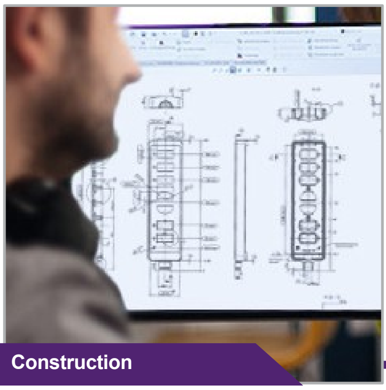
WHAT WE DO



We develop and manufacture customized products for various industries and provide our customers with comprehensive support from the initial idea through to series production. Our portfolio includes electromechanical input units and all components of electronic products, including housings, displays, keyboards and cable assemblies.

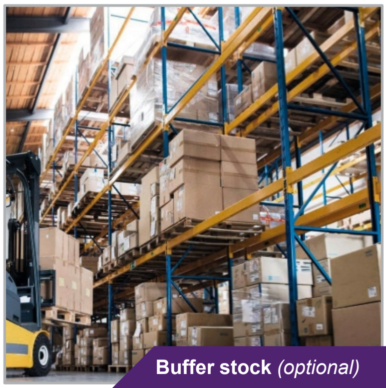
Our services range from advisory development and feasibility studies to cost estimates, prototype construction and material selection, right through to cost optimization and product design. We also create technical drawings and, if required, take on the complete design.

Engineering support from A-Z



Construction

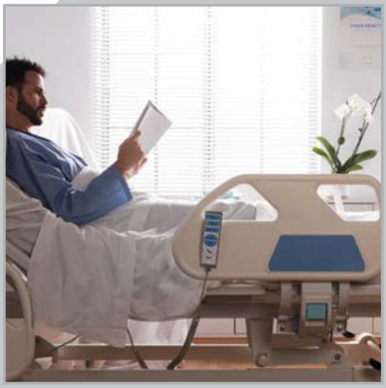
Procurement & Production



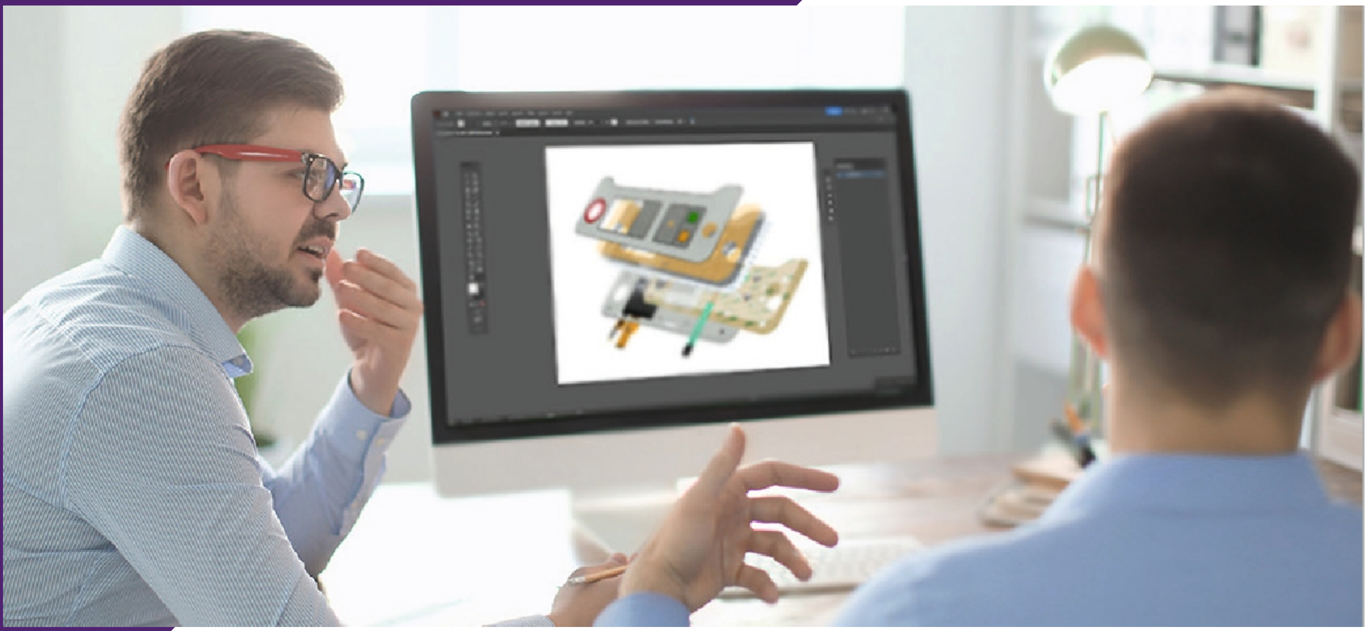
Buffer stock *(optional)*

Logistical processing

Assembly



Finished product



WHAT WE OFFER

Technical support

- Support from the concept phase to series development
- Feasibility studies
- Suggestions for improvement
- Advice on material selection and production methods
- Development of options to reduce costs

Development & Construction

- Development of components, molded parts, assemblies and complete solutions
- Sketching, conception and pre-construction
- Design in 3D CAD
- Optimization of existing customer templates
- Presentation of product views in the form of realistic 3D renderings
- Creation of production documents such as technical drawings and parts lists
- Prototype construction using 3D printing & silicone casting

N&H Laboratories

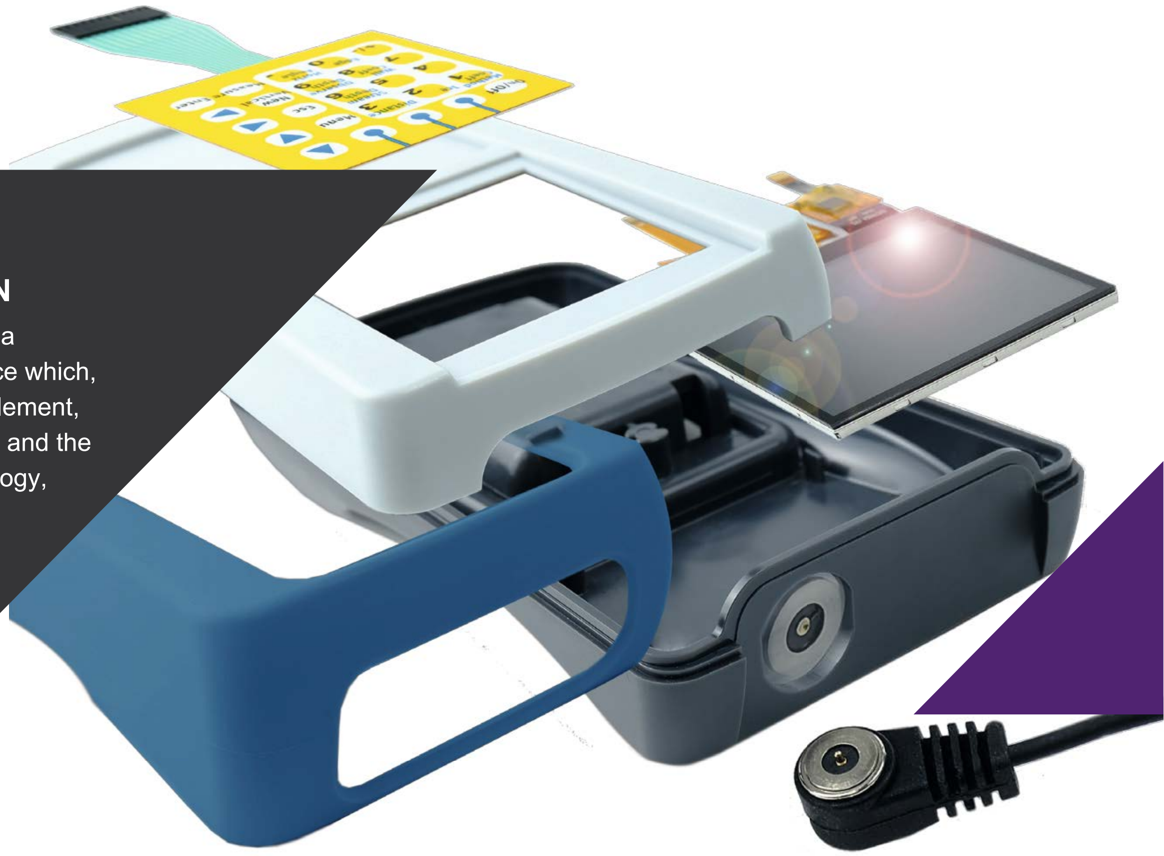
- Project-specific final inspection
- Electromechanical tests
- Optical / acoustic tests
- Material testing
- Measurements Surface resistance, Volume resistivity, conductivity
- Technical problem analysis, also for third-party products

Purchasing

- Outsourcing options for your supply chain
- Procurement of third-party components

Logistics

- Complete logistical handling
- Buffer stock at N&H Technology in Willich possible



COMPLETE SOLUTION

A typical product example is a customer-specific input device which, in addition to the keyboard element, includes a housing, a display and the complete connection technology, including cable assembly.

Customized components

KEYBOARDS

- Silicone rubber keypads
- Membrane keypads
- Capacitive keyboards
- Touch input systems

BUTTONS & SWITCHES

- Pushbutton
- Piezo push-button
- Status/signal lamps
- Micro switch

CABLE ASSEMBLY

- Cable harnesses
- Data cable
- Coaxial cable
- Special cable
- Individual cables

CONNECTORS

- Magnetic plugs
- Spring contact plug
- Special plug

PCB

- Flex & Rigid circuits
- Single layer, double layer, multilayer

FURTHER

- Protective bags
- Battery contacts

We also offer a wide range of standard components, which you can select & request directly from our **online catalog**!

katalog.nh-technology.de

Standard components

- Spring contacts / Pogo pins
- Plug connector
- High-current connectors

- Stainless steel keyboards
- Hygienic keyboards

- Micro switches for SMT
- Pushbutton, piezo pushbutton
- Status lamps

- LC displays (TFT)

- Transducer, buzzer
- Microphones, loudspeakers

Customized moulded parts

PLASTIC

- Precision & large parts
- Single and multiple spraying

ELASTOMER

- Protective covers
- O-rings, seals
- Precision parts

2K / 3K PARTS

METAL

- Heat sink
- Die-cast parts
- Stamped, turned and milled parts
- Deep-drawn parts
- Battery contacts

GLASS

- Front glasses
- Molded glass panes

DESIGN GUIDE

PLASTIC MOLDED PARTS

Specializing in custom plastic parts and assemblies, we utilize the versatility of plastics to create functional and aesthetically pleasing solutions. Our expertise in material selection and design allows us to integrate several key functions into each product.

We offer a comprehensive selection of plastics and granulates for the production of precision parts, housings and multi-component injection molded parts.

Our portfolio includes standard plastics, engineering plastics and high-performance plastics, which we process using state-of-the-art production techniques. This includes both classic injection molding and sophisticated multi-component injection molding.

We also offer a wide range of services. You can choose whether you want to make use of our development and design services separately or whether you want us to accompany you in all further steps up to series production.

Our experienced product designers are able to design individual components as well as complete products according to your wishes or to optimize the design and construction of existing products.

Our services in detail:

- Sketching, conception, planning and pre-construction
- Design in 3D CAD
- Presentation of product views in the form of realistic 3D renderings
- Creation of production documents such as technical drawings and parts lists
- Creating prototypes using 3D printing & silicone casting

ADVANTAGES OF PLASTICS

- **LIGHTWEIGHT**
  Plastics are significantly lighter than metals, which results in a reduction of the total weight.
- **STABILITY**
  Plastics are resistant to many chemicals and environmental influences. They absorb very little water and are therefore well suited for use in damp environments.
- **ISOLATION**
  Plastics have very good thermal and electrical properties. Insulating properties.
- **MECHANICAL PROPERTIES**
  Many plastics offer excellent strength, rigidity and impact resistance.
- **COLOR VARIETY AND DESIGN**
  The variety of colors and flexibility in design make plastics a popular choice for aesthetically demanding applications.
- **DURABILITY**
  Plastics have a long service life and retain their their properties over a long period of time.
- **RECYCLABILITY**
  Many plastics can be recycled and reused. which reduces their environmental impact.

DESIGN GUIDE

This Design Guide is aimed specifically at developers, designers and purchasers who deal with molded plastic parts.

It offers a comprehensive and detailed overview of engineering plastics and their diverse areas of application.

The correct selection and processing of these materials are decisive for the quality and functionality of the end products.

This guide places particular emphasis on the essential aspects of material selection and design, as well as the planning process that enables specific molded parts to be developed in close cooperation with our experienced engineers.

Our experts constantly **check and optimize** the design drawings to ensure that the end products meet the highest quality and functional requirements.

INTRODUCTION

TYPES OF PLASTIC

SUBDIVISION OF PLASTICS

Plastics can be divided into three main categories:

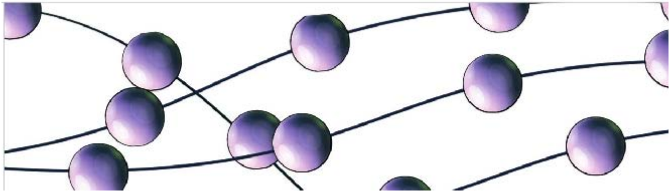
- **Thermoplastics** that become malleable when heated
- **Thermosets** that remain dimensionally stable after curing
- **Elastomers** that have a certain degree of flexibility

Further subgroups are based on origin (bio-based, petroleum-based), processing methods and specific properties.

Molded plastic parts, such as plastic housings, are an integral part of modern technical applications. By combining different materials, the specific advantages of each plastic can be optimally utilized.

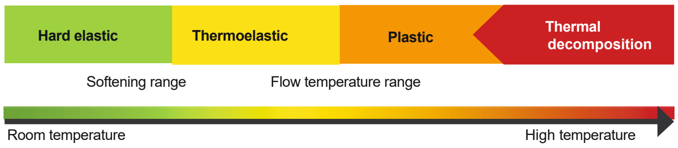
For example, polycarbonates (PC) are for areas that require high impact resistance and clarity, while polyamides (PA) are used in parts that require high strength and toughness.

Thermoplastics

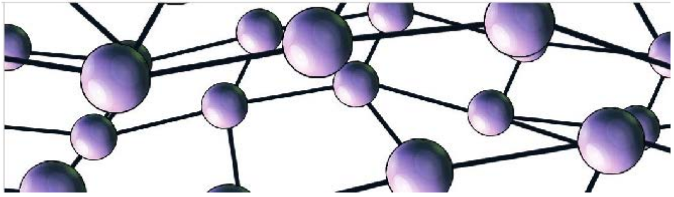


- Unbranched and long molecular chains (amorphous / semi-crystalline)
- Subdivision into standard thermoplastics, Engineering thermoplastics and high-performance thermoplastics
- Hot formable, forming repeatable
- Recyclable

Thermal behavior

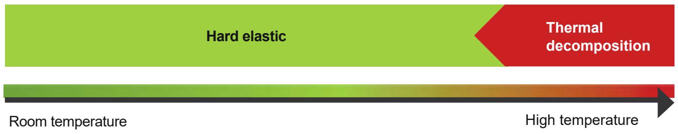


Thermosets

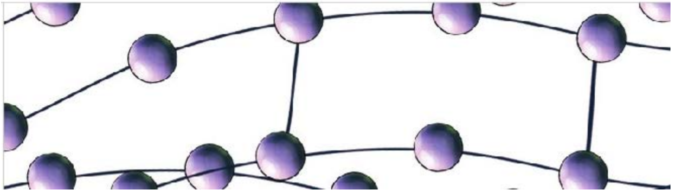


- Tightly cross-linked molecular chains
- Hard & brittle
- Temperature resistant
- Non-deformable, non-meltable
- Insoluble, not recyclable

Thermal behavior

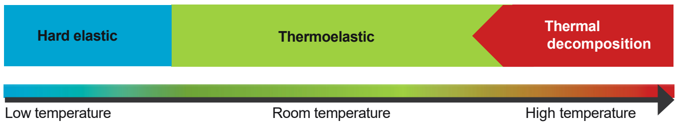


Elastomers



- Weakly cross-linked molecular chains
- Swellable, rubber-elastic
- Not recyclable

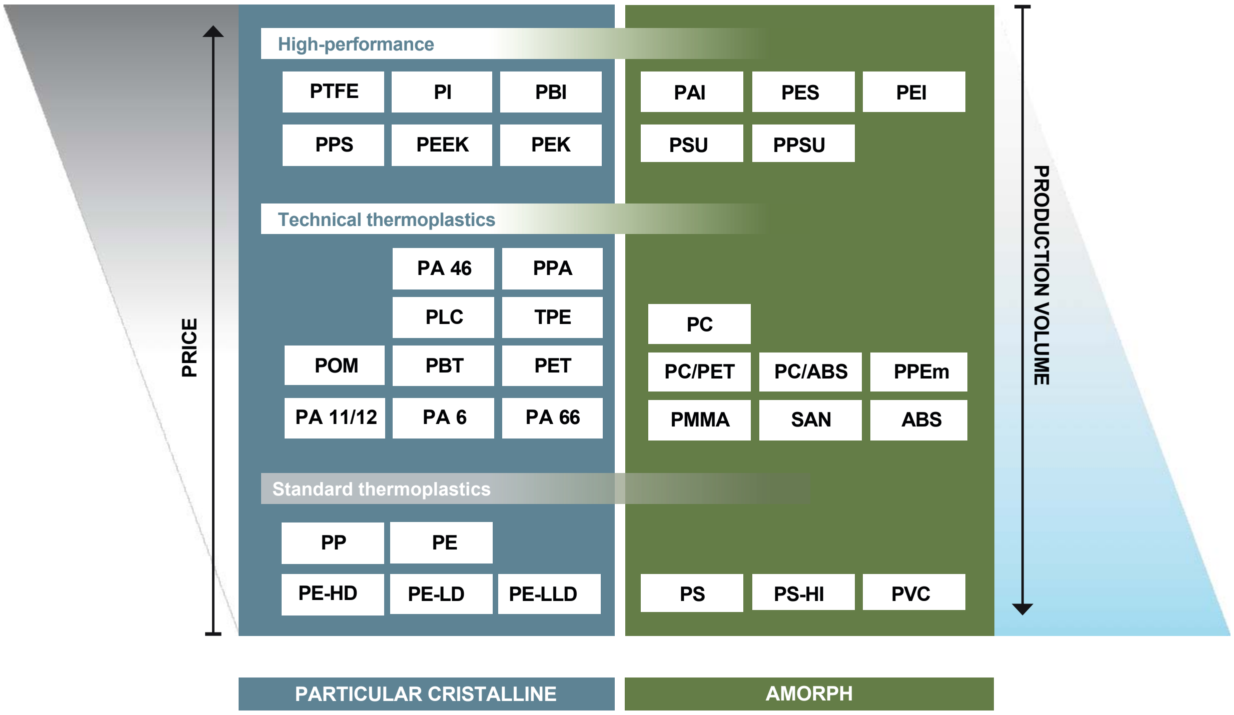
Thermal behavior



THERMOPLASTE

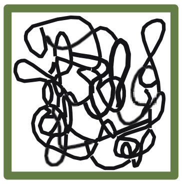
Thermoplastics become malleable when heated. In contrast to thermosets, which retain their shape permanently after curing, thermoplastics can be reshaped as often as required by applying heat.

The diagram shows the properties of various thermoplastics and illustrates the wide range of engineering plastics available. This means that a balanced price-performance ratio in relation to the planned production volume can be taken into account right from the start of planning.



Amorphous and semi-crystalline thermoplastics

Semi-crystalline thermoplastics form dense crystals when cooled. They are opaque, mechanically stronger, harder, tougher, resistant to chemicals and have a higher heat deflection temperature.



Amorphous thermoplastics have irregular molecular chains. They are transparent, brittle, hard and have glass-like properties in the solid state.

They also shrink less, are less prone to warping and have no fixed melting point. Their transparency makes them suitable as light guides, for example.

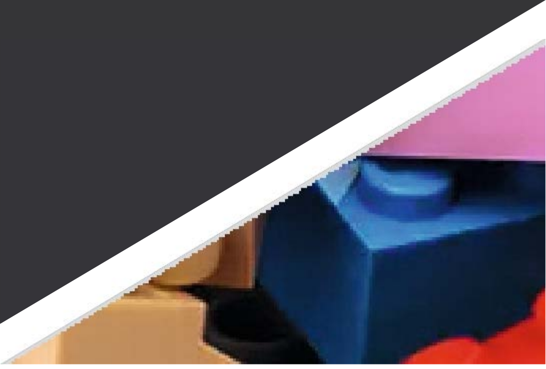
- **Standard thermoplastics** are versatile polymer materials,known for their balanced physical, mechanical and and thermal properties. They are easy to shape and easy to process, but less strong, hard and heat-resistant than technical or high-performance thermoplastics.

- **Technical thermoplastics** offer greater strength and rigidity. They are used in applications that require robust mechanical properties but do not place extreme demands on hardness or heat resistance.

- **High-performance thermoplastics** are specialized plastics with outstanding thermal and mechanical strength and high chemical resistance. They are used in demanding applications that require high strength, impact resistance, hardness and heat resistance, such as in the automotive and electronics industries.

PROPERTIES

THERMOPLASTE



THERMAL PROPERTIES

	ABS	**PC**	**PC GF 30**	**PA 6 G**	**PEEK**	**PEEK GF 30**	**POM-C**	**POM-C GF25**
Melting temperature (°C) Glass	110		235	215	340	340	165	165
transition temperature (C°) *Dynamic*		148		40	50	143	-60	-60
Thermal conductivity *Lambda λ at 23°C (W / (K-m))*	0,17	0,4	0,28	0,29	0,25	0,43	0,31	0,28
Coefficient of linear expansion *average value 23 - 60°C / (10*$^{-6}$*/K)*	95	65	40	80	50	30	110	
Heat distortion temperature (C°) *1.8 MPa measurement*	80	130	140	80	160	230	100	150
Upper service temperature *Short-term in air (C°)*	100	135	160	170	310	310	140	140
Upper service temperature *permanent, while 5,000 / 20,000*	- / 95	130 / 120	140 / -	105 / 90	- / 250	- / 250	115 / 100	100 / -
Lower service temperature *in air (C°)*	-30	-50		-30	-50	-65	-50	
Burning behavior *Oxygen index (%)*	19	25		25	35	40	15	
Burning behavior *Specific heat capacity J/(g - K)*	1,4	1,2	1,08	1,7	0,32		1,5	

MECHANICAL PROPERTIES AT 23°C

	ABS	**PC**	**PC GF 30**	**PA 6 G**	**PEEK**	**PEEK GF 30**	**POM-C**	**POM-C GF25**
Yield / breaking stress *MPa*	45 / -	74 / -	100 / -	86 / -	115 / -	OSP / 87	66 / -	80 / -
Tensile strength *MPa*		75		88	115	87	66	
Stretching elongation *%*		6		5			20	
Elongation at break / elongation at break *%*	10 / -	> 50 / -	- / 3	25 / -	17 / 25	3 / 2,5	50 / -	- / 3
Tension - Elasticity *N /* ^{mm2}	2.300	2.400	8.600	3.600	4.300	7.000	2.800	9.300
Nominal compression *1 / 2 / 5 %*		18 / 35 / 72		26 / 51 / 92	38 / 75 / -	54 / 103 / -	19 / 35 / 67	
Charpy impact strength *kj /* ^{m2}	22	9	7	3,5	3,5	3	7	30
Izod notched impact strength *kj /* ^{m2}		9,00		7,00			7,00	
Ball pressure hardness *N /* ^{mm2}	90	120	145	165	190	215	140	
Rockwell hardness *N /* ^{mm2}	R 104	M 75		M 88	M 105	M 100	M 84	
Coefficient of sliding friction *μ*		0,5 - 0,6		0,4	0,30 - 0,50	0,3 - 0,45	0,3 - 0,45	0,5 - 0,6

PHYSICAL PROPERTIES

	ABS	**PC**	**PC GF 30**	**PA 6 G**	**PEEK**	**PEEK GF 30**	**POM-C**	**POM-C GF25**
Density (g/cm³)	1,04	1,2	1,42	1,15	1,31	1,51	1,47	1,58

ELECTRICAL PROPERTIES

	ABS	PC	PC GF 30	PA 6 G	PEEK	PEEK GF 30	POM-C	POM-C GF25
Dielectric strength *kV / mm*	41	28	30	25	24	24	20	50
Contact resistance *Specific (Ω-m)*	10^{12}	10^{13}	10^{14}	10^{12}	10^{12}	10^{13}	10^{13}	10^{14}
Surface resistance *Specific (Ω)*	10^{15}	10^{15}	10^{16}	10^{13}	10^{13}	10^{14}	10^{13}	10^{14}
Dielectric constant Epsilon ε *at 100 Hz*		3,00		3,60	3,20	3,20	3,80	
Dielectric constant Epsilon ε *at 1 MHz*	2,6	3	3,2	3,2	3,2	3,6	3,8	4,8
Dielectric constant Dissipation factor *at 100 Hz (tan delta)*		0,0010		0,0120	0,0010	0,0010	0,0030	
Dielectric constant Dissipation factor *at 1 MHz (tan delta)*	0,008	0,008	0,008	0,016	0,002	0,002	0,008	0,005

WATER ABSORPTION

	ABS	PC	PC GF 30	PA 6 G	PEEK	PEEK GF 30	POM-C	POM-C GF25
after 24/96 h storage *in water at 23°C (mg)*		13 / 13		44 / 83	5 / 10	5 / 10	20 / 37	
after 24/96 h storage *in water at 23°C (%)*		0,18 / 0,33		0,65 / 1,22	0,06 / 0,12	0,05 / 0,10	0,24 / 0,45	
at saturation in a normal climate *23°C / 50% RF (%)*	0,22	0,15	0,10	2,20	0,20	0,16	0,20	0,20
when the water is saturated *23°C (%)*	1,00	0,40	0,28	6,50	0,45	0,35	0,80	0,6

CHEMICAL RESISTANCE (excerpt)

	ABS	PC	PA	PEEK	POM
Acetone	–	–	+	+	+
Ammonia	25%	–	10%	+	+
Gasoline	–	△	+	+	+
Brake fluid	△	–	+	+	+
Calcium chloride	+	+	10%	+	+
Chlorobenzene	–	–	+	+	+
Diesel oil	+	△	+	+	+
Acetic acid	25%	10%	5%	△	+
Formaldehyde	30%	40%	△	+	+
Glycerine	+	△	+	+	+
Heating oil	△	△	+	+	+
Potassium hydroxide	50%	–	50%	+	+
Potassium chloride	+	+	10%	+	+
Methanol	△	–	△	+	+

	ABS	PC	PA	PEEK	POM
Methylene chlorid	–	–	△	+	–
Lactic acid	80%	+	△	△	+
Engine oils	+	+	+	+	+
Sodium carbonate	+	+	10%	+	+
Sodium chloride	+	+	–	+	+
Sodium hydroxide	+	+	+	+	+
Caustic soda	50%	50%	+	+	+
Nitric acid	10%	–	–	+	–
Hydrochloric acid	10%	5%	–	△	–
Sulphuric acid	50%	50%	–	△	+
Turpentine	–	–	+	+	+
Trichloroethylene	–	–	+	+	△
Tartaric acid	+	+	10%	+	–
Xylene	–	–	+	+	+

consistent	+
resistant to max. % concentration	%
conditionally stable	△
not stable	–

We will be happy to send you a complete list of chemical resistance on request.

PLASTIC MOLDED PARTS

DESIGN PRINCIPLES

Methods for optimizing durability, functionality and aesthetics are presented below. These take into account the flow behavior of plastics, material selection and tool design. Efficient mold design enables cost-effective production and can improve product life and performance.

DESIGN FOR PRODUCTION AND ASSEMBLY

Design for manufacturing and assembly (DFM) is a decisive approach in the development and production of molded plastic parts. DFM offers considerable advantages for users in the field of molded plastic parts. For example, the application of DFM principles increases the efficiency of production processes and product quality.

Integration also means closer collaboration with the design and production teams, enabling buyers to negotiate better with suppliers and source more cost-effective, high-quality parts.

Finally, the DFM process supports sustainability in production. By reducing the variety of materials and using recyclable materials, waste and environmental impact are minimized.

ADVANTAGES OF DFM

Cost reduction

- Optimized design
- Less waste
- Fewer errors
- Fewer revisions

Time saving

- Streamlined process
- Shorter production lead time
- Higher production efficiency

Quality improvement

- Consistent results
- Greater innovation

DFM PROCESS

PROCEDURE

The first step is to select the right manufacturing process for the component.

DESIGN

This is followed by the design of the component. This is drawn on the basis of the principles specific to the selected manufacturing process.

MATERIAL

The factors that play a role in material selection include mechanical, thermal, optical, electrical and fire-retardant properties as well as color.

ENVIRONMENT

The components must be designed in such a way that they can withstand the environment in which they are used.

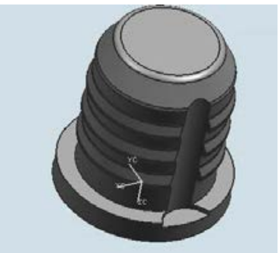
CONFORMITY

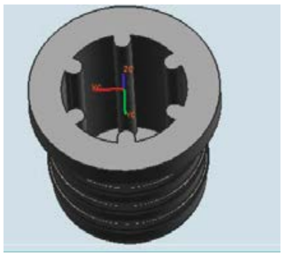
Product & components must comply with quality and safety standards.
These standards may include industry standards as well as company or third-party standards.

N&H - DFM REPORT

Example of the contents of a DFM report that we create for our customers.

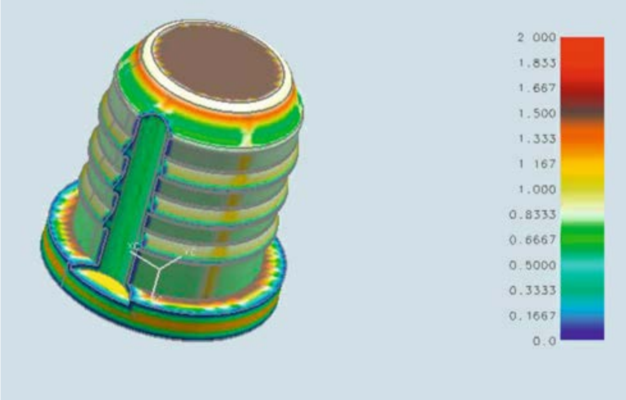
Component information



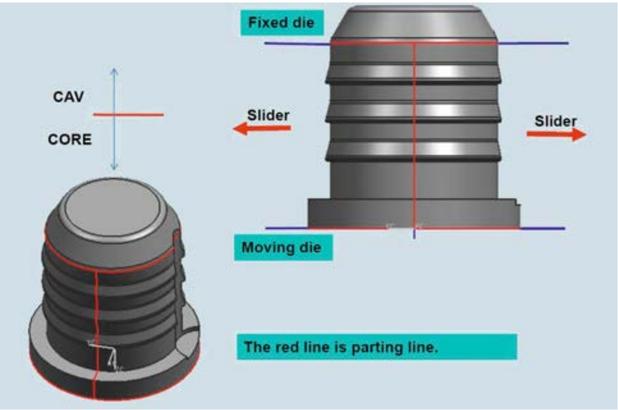


N&H No.	DTIO-53540-B0	Part Name	Bushing L2
Unit of Set	8pcs / set	Forecast	~16k / year
Material	TPE ShA70	Color	black
Texture	TBD	Finish	none
Outlook Size	D11x11,5mm	Weight	NW 0.44g

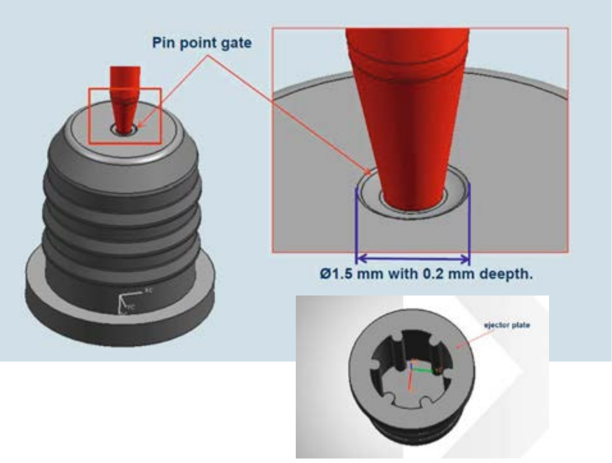
Wall thickness



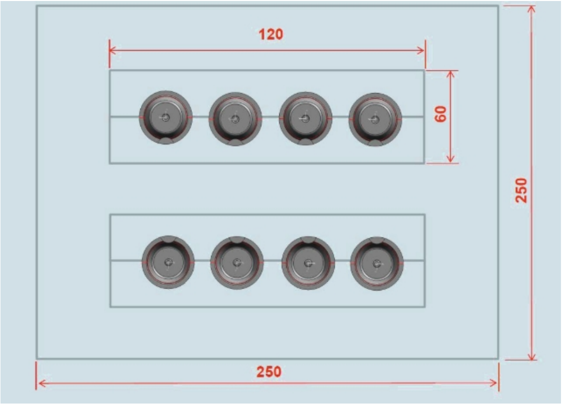
Separating plane



Injection point



Cavity layout



DESIGN PRINCIPLES

BASIC RULES

Here you will find an overview of the basic design rules that should be observed when developing plastic parts. Adhering to these principles not only makes the manufacturing process easier, but also contributes to the production of high-quality and functional products.

CHECKLISTE

- ☐ Keep wall thicknesses as low as possible
- ☐ Maintain uniform wall thicknesses
- ☐ Avoid mass accumulation
- ☐ Round off corners and edges
- ☐ Designing ribs for spray application
- ☐ Avoid flat surfaces
- ☐ Provide draft angles (tapers)
- ☐ Avoid undercuts
- ☐ Integrate functions (e.g. film hinges or hooks)
- ☐ Use the option of tool-based design
- ☐ Avoid complex sliders
- ☐ No excessive tolerance requirements

TOLERANCES

Plastics are physically very different from metals, in particular due to their higher thermal expansion, volume changes due to moisture absorption and dimensional changes over time due to residual stress reduction. These properties make it difficult to maintain tight dimensional tolerances, which is why plastic components should be toleranced differently to metal components.

DIN ISO 20457 for plastic

DIN ISO 20457 provides guidelines for the definition of tolerances and the acceptance conditions for dimensions of plastic parts produced by various shaping processes such as injection molding, extrusion or thermoforming. For injection molding, the tolerance group TG 6 generally applies to thermoplastics.

DIN ISO 3302 for rubber

The tolerances for molded rubber parts are based on DIN ISO 3302, according to which molded parts are divided into four tolerance classes, ranging from M1 (fine) to M4 (coarse). Tolerance class M3 (medium) is generally used for technical molded rubber parts.

Depending on the orientation during the pressing process, the standard distinguishes between dimensions that are linked to the mold (F) and dimensions that are linked to a 2-component bonding system (C).

Deviations from the nominal dimension and subsequent machining generally depend on the respective manufacturing or production process. If the parts are machined, DIN ISO 2768 always applies.

DIN ISO 2768 - General tolerances

The DIN ISO 2768 standard is an internationally recognized standard that defines general tolerances for dimensions without specific tolerance specifications.

The standard is divided into two main parts:

DIN ISO 2768-1: This sub-standard covers general tolerances for linear and angular dimensions and is applicable when the length is measured directly between two points. It provides tolerance values for different classes - fine, medium, coarse and very coarse - depending on the precision requirements of the parts.

DIN ISO 2768-2: This part deals with general tolerances for shape and position, such as straightness, roundness, symmetry, etc. Various degrees of accuracy are also provided here according to the requirements.

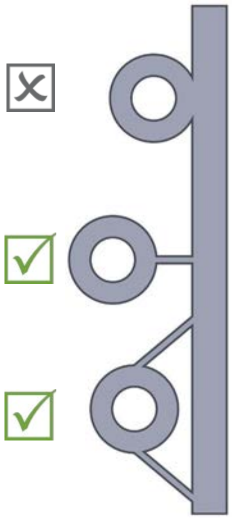
For mechanical engineering parts, the **degree of accuracy m (medium)** should be selected, in special cases f (fine).

AVOID MASS ACCUMULATION

Mass accumulations lead to various problems in the manufacturing process and also impair the quality of the end product. Among other things, they cause uneven cooling, which can promote internal stresses and distortion. Material shrinkage can also lead to visible sink marks or indentations on the surface.

Accumulations also make it difficult to distribute the material evenly, which can lead to defects such as air pockets or incomplete fillings. As thicker material areas require longer cooling times, the entire production cycle is extended. In addition, different material thicknesses can result in areas with weaker mechanical properties.

This should therefore taken into account when designing plastic molded parts,
Avoid material accumulation and strive for a uniform wall thickness.



Example: To avoid mass accumulation
eyes should not be connected directly to the wall.

WALL STRENGTH

Optimizing the wall thickness is crucial in the development and production of plastic moulded parts, as it ensures a balance between strength and material consumption.

Walls that are too thick increase material costs and cycle times, while walls that are too thin cause weak points that can affect the durability of the part. Uniform cooling is also important to avoid deformation and stresses.

Different plastics, such as ABS or polycarbonate, have different flow properties and shrinkage rates, which must be taken into account when determining the wall thickness.

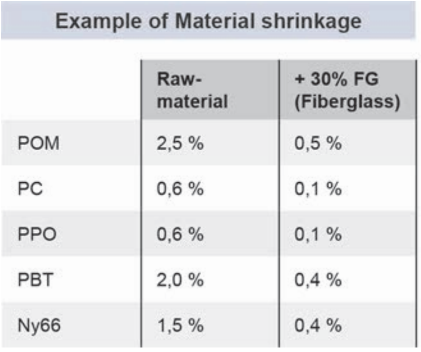
Wall thickness recommendations

	min. (mm)	max. (mm)
ABS	1,143	3,556
PC	1,016	3,180
Acrylic	0,635	3,810
Nylon	0,762	2,921
Polyester	0,635	3,175
PUR	2,032	19,05
Polyethylene	0,762	5,080
Polypropylene	0,635	3,810
Polystyrene	0,889	3,810

MATERIAL SHRINKAGE

Every plastic has a specific shrinkage behavior that must be taken into account right from the start of planning. This refers to the geometric change of a molded part during cooling from a molten to a solid state. This affects the dimensional accuracy and visual quality of the parts.

Countermeasures include the adaptation of tool designs and the optimization of injection moulding processes. Shrinkage can also be influenced by the addition of additives such as glass fibers. Our engineers will be happy to advise you on this.



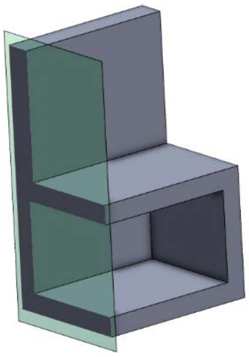
Example of Material shrinkage

	Raw-material	+ 30% FG (Fiberglass)
POM	2,5 %	0,5 %
PC	0,6 %	0,1 %
PPO	0,6 %	0,1 %
PBT	2,0 %	0,4 %
Ny66	1,5 %	0,4 %

DESIGN PRINCIPLES

CONSTRUCTION

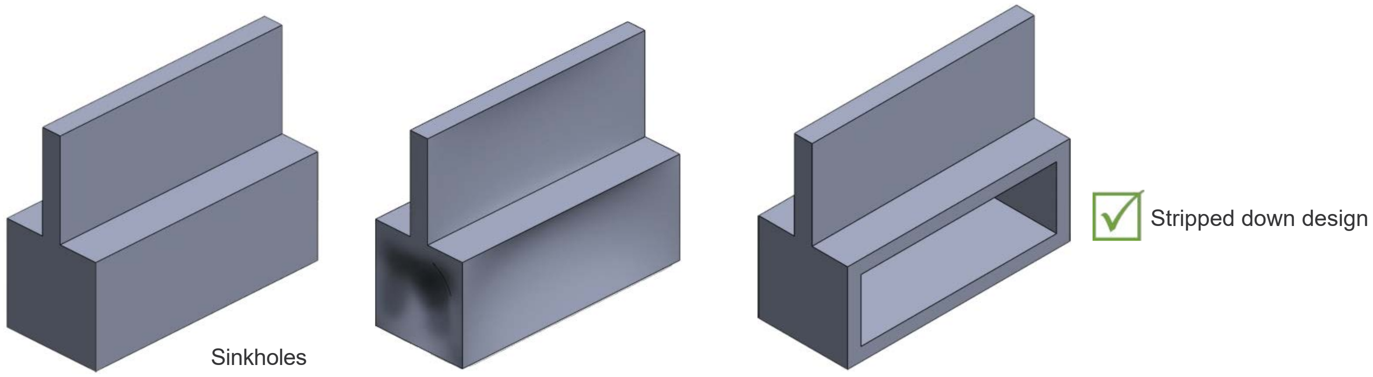
CORED GEOMETRY

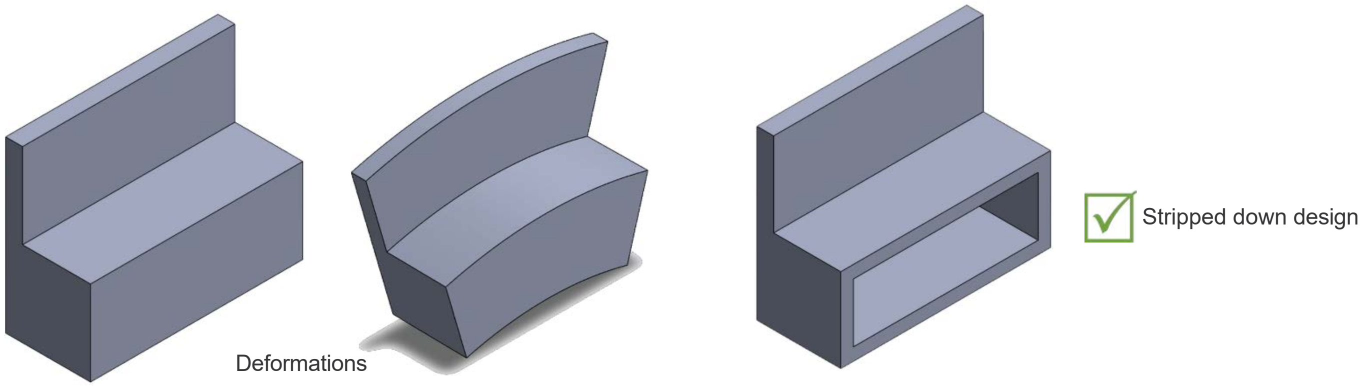


In order to avoid deformations and sink marks, coring the geometry is a proven method. A cored design allows the visual appearance to be retained, while the finished molded part is optimally designed with regard to all design criteria.

If a plastic molded part with seemingly simple geometry is manufactured as a solid piece of material, sink marks and deformations are very likely. The cored design prevents these or cleverly hides them inside in places that will not be visible later.

Please note that the adjacent material thicknesses must match. Otherwise, sinking can also occur.





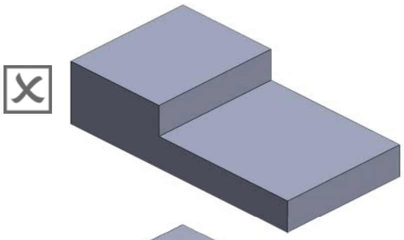
TRANSITIONS

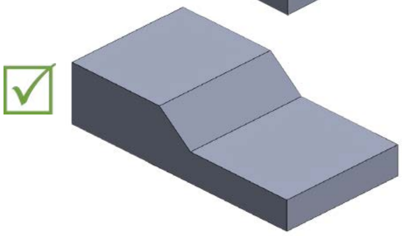
Avoidance of sharp transitions

It is important to avoid sharp-edged transitions, as these can cause casting-related stresses. In the worst case, this could result in an avoidable fracture.

The following example shows what appears to be the simplest form: right angle with the same leg thickness. In practice, however, there is a high probability that the sharp angle will create an unwanted predetermined breaking point.

Radii are much better suited to creating a stable angle. It is possible to make the radius relatively small, but the manufacture of the tool must be taken into account. The radius is milled in stainless steel and is subject to the manufacturing possibilities.





ROUNDED CORNERS

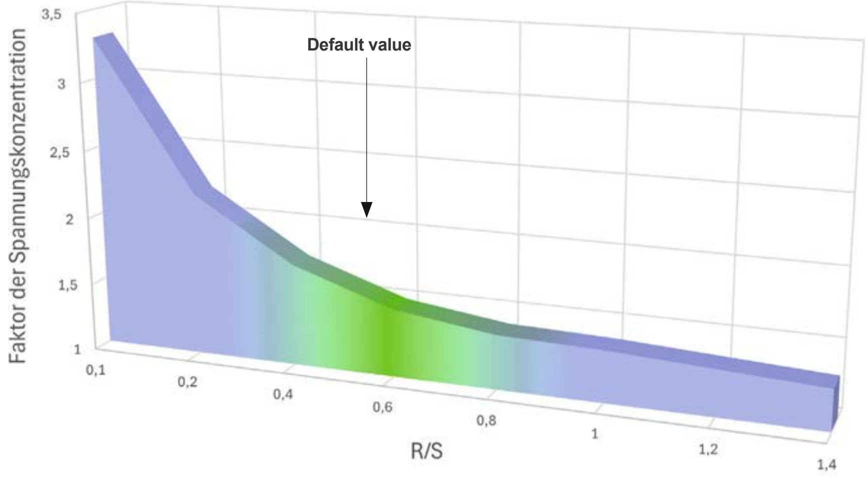
Avoidance of tension due to sharp edges

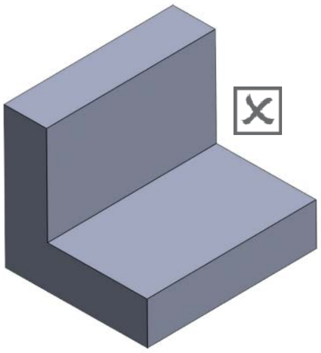
Sharp edges can cause stresses in the plastic during production as the material flow is impaired. To avoid this, rounded edges and smooth transitions between different wall thicknesses should be used.

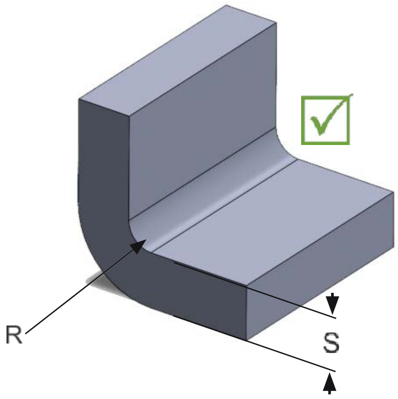
These design measures help to minimize stress concentrations
and to optimize the flow of the plastic during injection moulding.
The inner radius should at least correspond to the wall thickness.

If sharp corners are desired in the design, the inner corners should have radii.

Increasing the radii= Reducing the stress concentration





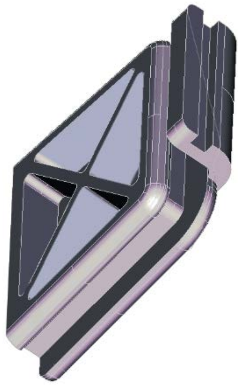


CONSTRUCTION

RIBS

Use of ribs to increase stability

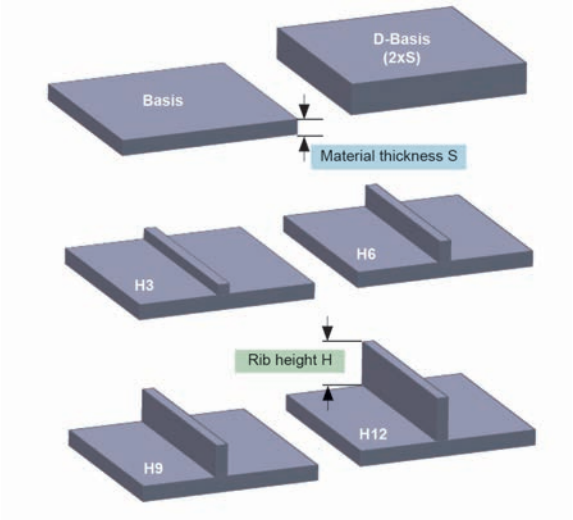
Ribs are an effective way of increasing stability, reducing material consumption and and to manufacture cost-effectively.



> **The correct design of ribs comprises five aspects: Thickness, height, position, quantity & malleability.**

The height of the ribs can increase the stiffness of the injection molded part in a geometric progression without much weight. For example (H12), if the rib height is increased to 12mm, the stiffness increases by 7.6 times while the weight only increases by 15%. However, if the total wall thickness S is doubled (D-basis), the weight of the material also doubles, but the stiffness only increases by a factor of 7.

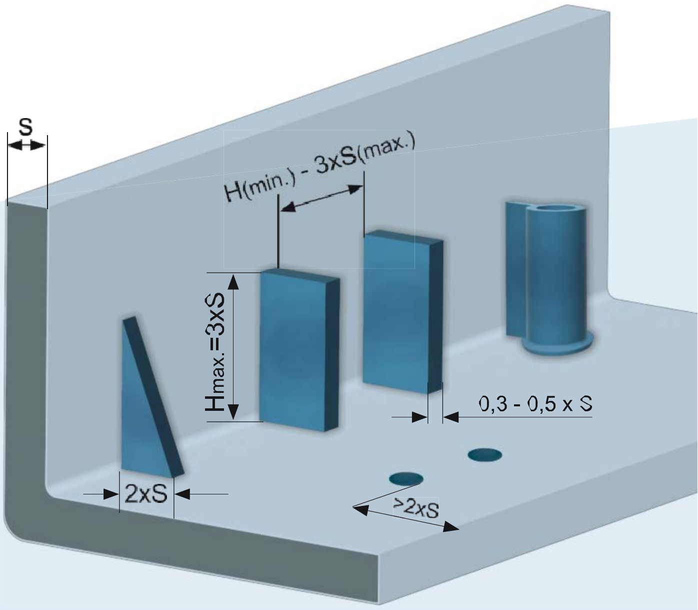
Model	**Base**	**H3**	**H6**	**H9**	**H12**	**D-base**
Material thickness S (mm)	4	4	4	4	4	8
Rib height H (mm)	0	3	6	9	12	0
Rib thickness W (mm)	0	2	2	2	2	0
Moment of inertia	213	289	528	1018	1837	1707
Increased rigidity (%)	0	35	148	377	**761**	**700**
Increased weight (%)	0	3,75	7,5	11,5	**15**	**100**



However, a high rib height also causes problems. **If the rib height is too high**, **filling problems** can during injection molding. If additional rigidity is required, it is preferable to use several shorter ribs.

When constructing ribs, it is important to consider the ratio of rib thickness to adjacent wall thickness.

Ideally, the rib thickness should be 40-60% of the adjacent wall thickness. Otherwise, sinking or demolding difficulties may occur.

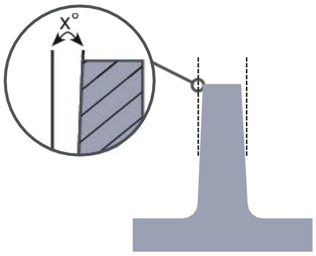


This equation can be used as a guideline:

Rib height:	H max. = 3 x S
Rib width:	For structured surfaces: 0.5 x S For polished surfaces: 0.3 x S

Demolding chamfers

The draft angles of ribs vary depending on the material and rib size. As a rule, it is 0.5° to 2.0°, with angles of 1.0° to 1.5° being used most frequently.



Material	Rib < 25mm	Rib > 25mm
PBT	0° - 0,25°	0,5°
PBT GF	0,5°	0,5° - 1,0°
PA	0,125°	0,25° - 0,5°
PA GF	0,2° - 0,5°	0,5° - 1,0°
POM	0° - 0,25°	0,5°

Demolding chamfers for ribs on thermoplastic components

Rib radii

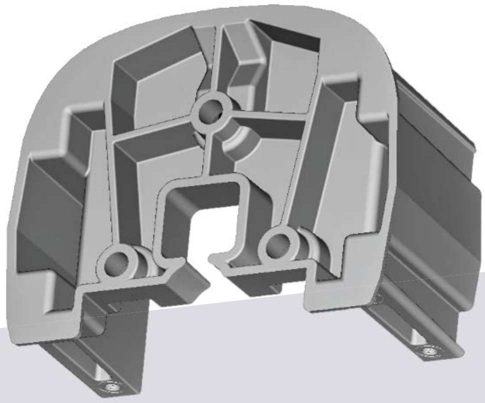
To reduce stress concentrations to, increase the strength of the ribs to and ensure a more even flow of the plastic material during injection molding, a radius should be taken into account at the connection between rib and base or side wall. It is recommended that these **radii** are **around 25 % to 50 %** of the nominal wall thickness.

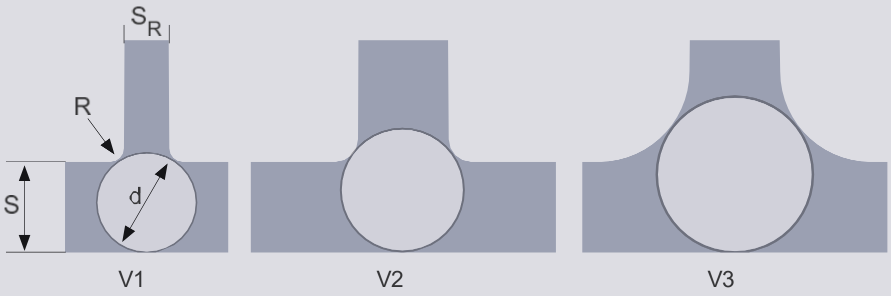
General recommendations

When planning the number and position of ribs, make sure that they do not unintentionally worsen the intended improvements. For example, ribs that are intended to reinforce and prevent fractures can impair the shock absorption of the component. A dense network of ribs can also make it more difficult for the mold to cool down and cause distortion.

From a design perspective, aligning the ribs along the bending force increases the rigidity of the component. From a production point of view, the ribs in the direction of the material flow minimizes the risk of incomplete filling and the formation of air pockets. If high rigidity is required on both sides, ribs should be arranged on both sides of the component.

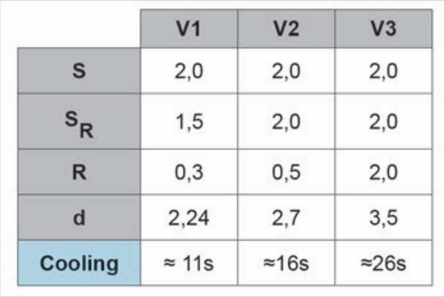
As it easier to add ribs than to remove them, they should be used with restraint during the initial shaping. If necessary, additional ribs can be added later for fine adjustment.





Influence of the rib thickness (S) and the rounding radius (R) on the cooling time.

The diameter (d) of the control circuit determines the maximum wall thickness that must be cooled.



	V1	V2	V3
S	2,0	2,0	2,0
S_R	1,5	2,0	2,0
R	0,3	0,5	2,0
d	2,24	2,7	3,5
Cooling	≈ 11s	≈16s	≈26s

DESIGN PRINCIPLES

CONSTRUCTION

DOMES / PROJECTIONS

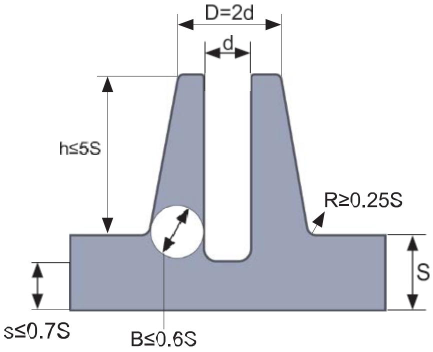
Optimization of screw domes

Screw bosses or bosses often take on functions for assembly in a plastic molded part. They support self-tapping screws, inserts or other fastening elements in a plastic part.

To avoid sinking, their thickness should be between 40% and 60% of the nominal wall thickness.

It should be noted that with thick, tower-like screw domes, indentations often occur on the back, which are visible. A better solution is to use thinner ribs and/or gussets.

Design guideline for screw domes



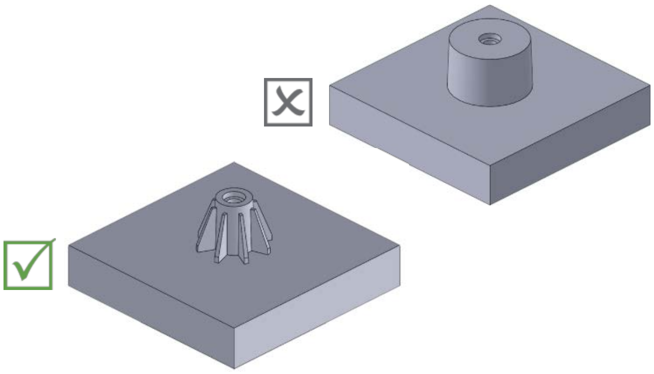
AMPLIFICATIONS

Reinforcements to prevent deformation

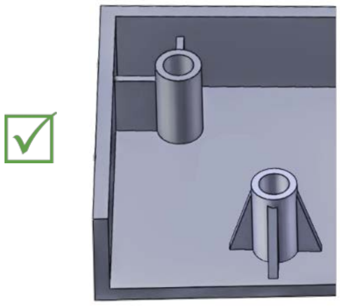
To counteract deformations, it is necessary to plan reinforcements skillfully. This makes angles more precise and ensures subsequent stability.

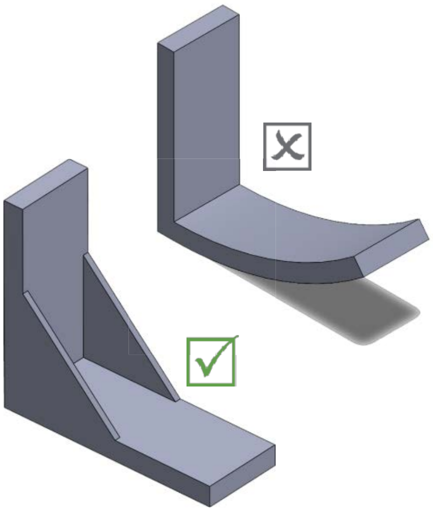
The shrinkage rate is lower in the vertical direction than in the flow direction, which leads to stresses in the molded part and thus to twisting and warping.

Reinforcements such as ribs can absorb these stresses and prevent deformation. Such reinforcements are often placed on the inside or in areas of the molded part that are not visible.



Always preferably use ribs or gussets to attach projections to a side wall or the floor.





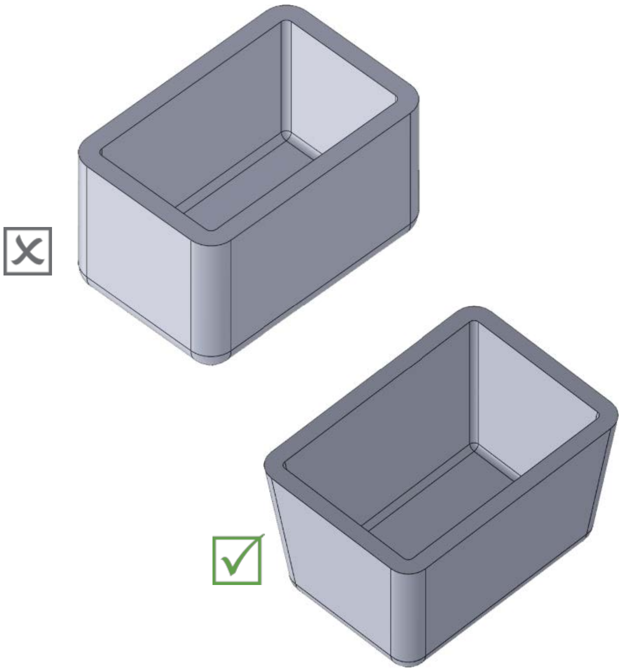
DRAFT ANGLES

Avoidance of ejector marks due to mold bevels

Ejector marks can be reduced or avoided altogether if chamfers are planned from the outset. If possible, all vertical surfaces should be beveled. Plan approximately one degree of bevel per 50 mm depth. Undercuts are possible, but require adjustments to the tool.

Guide values for draft angles

Material	Reference value
PA, POM, ABS, PP	0,5°
PBT, SB	1,0°
PS, PC	1,5°



UNDERCUTS

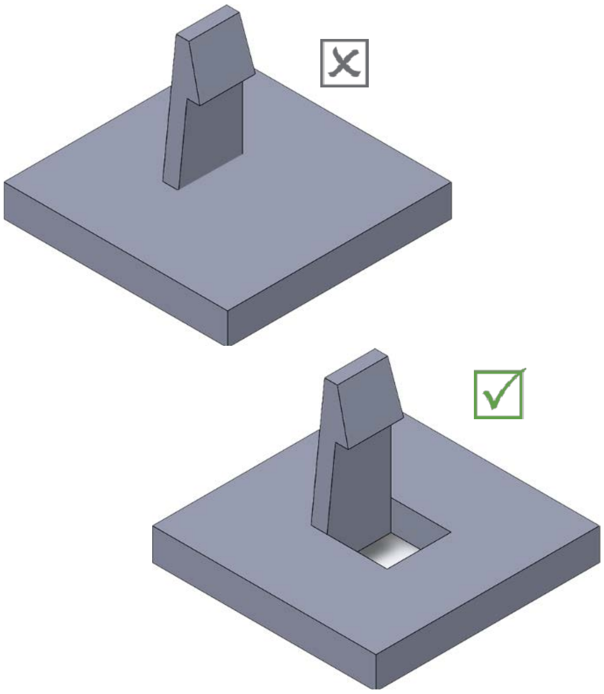
Alternative solutions to tool slides

Normally, undercuts are realized by using slides, which significantly increases the cost of the tool.

However, there are more cost-effective alternatives, such as inserting additional recesses or adjusting the parting line to optimize the demoulding angle.

These methods are particularly suitable for undercuts on the outside of the part. In some cases, the undercut can also be overcome by slightly deforming the flexible part during demolding. For internal cores, it is important to consider a demolding angle of 30° to 45° to ensure smooth ejection.

If these cost-effective solutions are not applicable and the functionality of the part must not be impaired, sliders and lateral cores must be planned.



CONSTRUCTION

FILM SHARING

Film hinges are flexible, thin-walled connections between two components that can be manufactured as a one-piece component in a single injection molding process. Preferred materials for film hinges are polypropylene (PP), while ABS, POM, PA and other plastics can also be used for applications with lower load cycles.

For a hinge with a constant thickness, the occurring edge fiber elongation can be calculated using a formula. The edge fiber elongation must be compared with the elongation value that can be tolerated by the desired number of load cycles (table), whereby a safety factor of 1.5 to 2 should be taken into account. This comparison a statement about the functional suitability of the film hinge.

The equation can be rearranged to calculate the film thickness s or film length L by selecting the plastic and inserting the value of the tolerable edge fiber elongation (with safety margin).

Using the formulas and the table as an example, you can, for example calculate the length of a film hinge (thin section L):

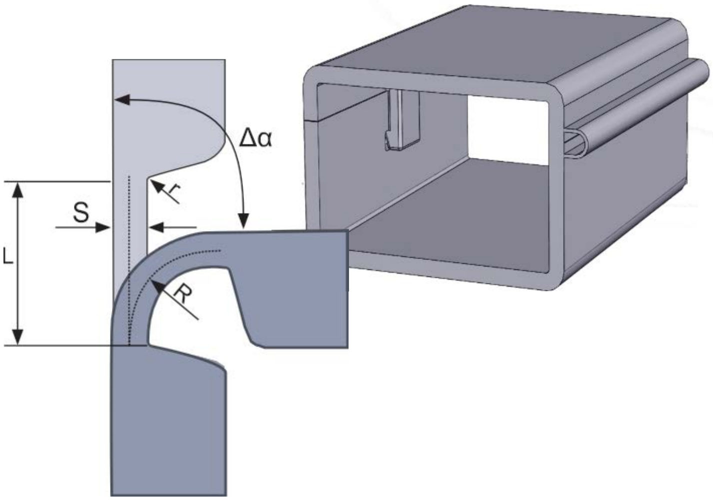
- Bending angle of at least 100°
- 10^4 Load change (LW)
- Thickness of the thin section= 0.3 mm
- Safety factor = 1.5
- Material= POM

L = (0.3 x π x 100 x 1.5) / (2 x 35) = 2.0mm

The conflicting requirements for film hinges - high mobility on the one hand and good injection moldability on the other - make a compromise necessary.

The following dimensions are recommended:

Film thickness s = 0.3mm to 0.8mm Film length L = 1mm to 6 mm Rounding radii R= 0.5mm to 1.0 mm



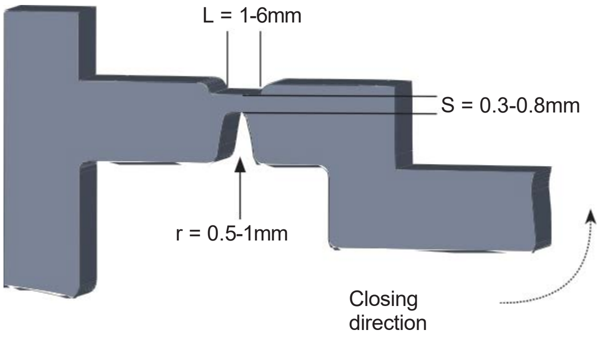
Randfaserdehnung ε_a $\quad \varepsilon_a = \frac{s \cdot \Delta a_{rad}}{2 \cdot L}$ mit $\Delta a_{rad} = \frac{\pi}{180} \cdot \Delta a°$
Edge fiber stretch

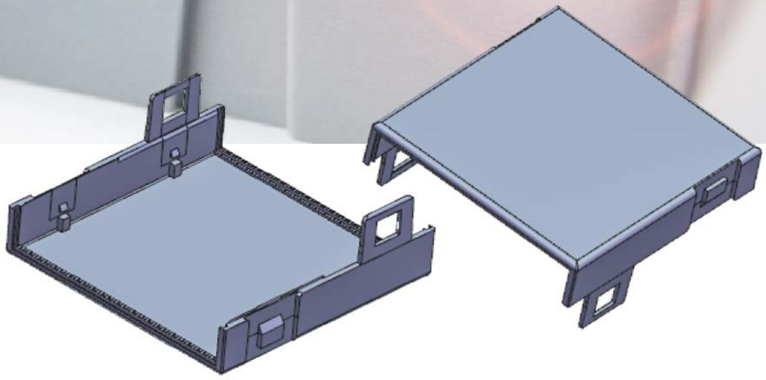
Länge L der Dünnstelle $\quad L = \frac{s \cdot \Delta a}{2 \cdot \varepsilon}$
Length L of the thin spot

Dicke s der Dünnstelle $\quad s = \frac{2 \cdot n \cdot \varepsilon}{\Delta a}$
Thickness D of the thin spot

Material	Stretching elongation εγ in %	Nominal elongation εt in %	Expansion deflection εa in % after 10^4 LW	Strain deflection εa in % after 10^5 LW
PP	10	> 50	60	60
PA 6	20	> 50	55	45
PA 66	20	> 50	50	40
POM	10	35	35	30
PBT	3,5	> 50	25	20

Strain values measured according to ISO 527, and as component-specific values measured on film hinges. Wöhler diagrams are usually used for the dimensioning of frequently actuated film hinges.





SNAP HOOKS

Snap hooks offer a simple, cost-effective and quick method of joining two different components together. During the joining process, the hook is briefly deformed and snaps into a recess (undercut) in the opposite part. After joining, the snap connections should return to their tension-free state.

There are different types of snap-fit connections, such as cantilever, torsion and ring-shaped snap-fit connections, each of which offers different application possibilities. Depending on the design of the undercut, the connection can be either detachable or permanent, whereby the force required to separate the parts can vary greatly.

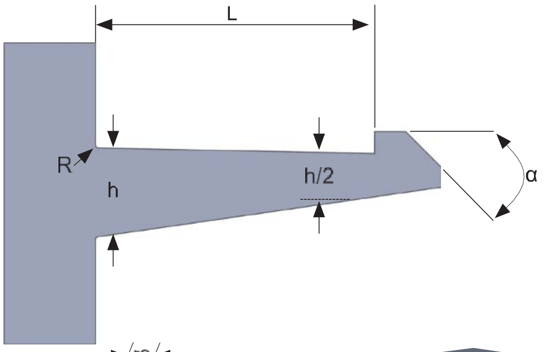
When designing snap-on connections, the following factors must be taken into account:

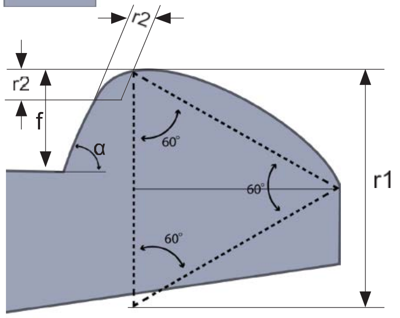
The thickness (h) of the hook should taper towards the end. It is good practice to reduce the thickness of the boom linearly so that at the end of the hook (length L) it is approximately half the wall thickness (h/2). This ensures a balanced ratio of cross-section and load, reduces material stress and saves material.

An **optimized geometry** of snap-fit hooks uses large and small radii for the transitions instead of inclined planes with fixed angles on the joining and holding sides. This design almost halves the force required for joining and reduces the pressure on the contact surfaces by up to 75 %.

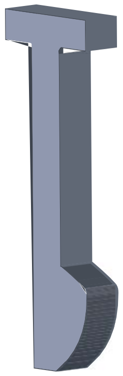
Note:
There are specific calculation formulas for the design and dimensioning of snap-fit connections, which determine important parameters such as release force, maximum load and deformation.

Detailed formulas and calculation methods can be found in technical textbooks on plastics technology.

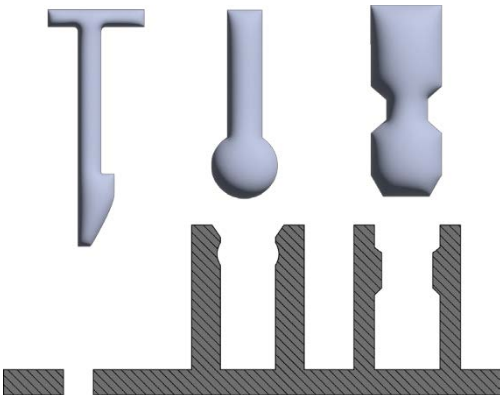




Optimized geometry



Cantilever, ring and torsion snap connections



PLASTIC MOLDED PARTS

TOOL CONSTRUCTION

Plastic injection molding is an important manufacturing method for the rapid production of complex plastic parts. The injection mold is particularly important in 2K injection molding, where two different plastics are combined in a single process. The spectrum ranges from simple two-part molds to complex tools with multiple slides. The complexity of the mold influences both the production speed and the costs.

Both the filling and demoulding of the molded part should be taken into account at the design stage. A well thought-out design can significantly reduce mold costs.

An injection mold usually consists of two mold halves: the nozzle side and the ejector side (2-plate mold). With the 3-plate mold, an additional plate ensures that the sprue is automatically separated from the molded part.

The injection molding process comprises the following steps:

- **Plasticizing & dosing**
- **Injection & pressurization**
- **Cooling & demolding**

The plasticized plastic is injected into the mould via the nozzle-side half of the mould. The sprue is selected according to the component geometry. *(see chapter gate types)*

The ejector side of an injection mold contains the shaping cores and inserts, also known as cavities, as well as the ejector elements.

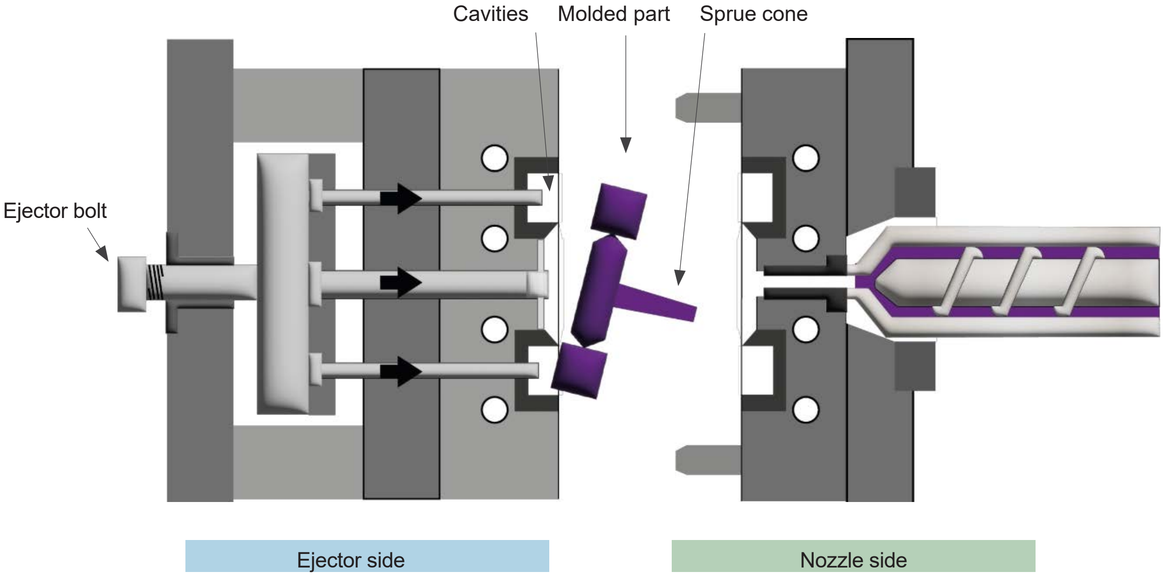
When the mold is opened, the molded part usually remains on this side. Different ejector devices are used depending on the degree of difficulty of demolding.

EXTERNAL SYTEME

Injection molded parts without undercuts can be stripped, removed or demolded with ejector pins.

Various ejector pins are available:
Cylindrical ejectors, flat ejectors and contour ejectors.

The ejectors should be placed in such a way that they push the part evenly out of the mold. Critical areas such as thin walls or surfaces that require a high aesthetic quality should be protected from pressure marks.



In general, the **ejection force** must be carefully adjusted so as not to damage the molded part. It depends on the adhesion of the plastic to the mold and the size of the molded part.

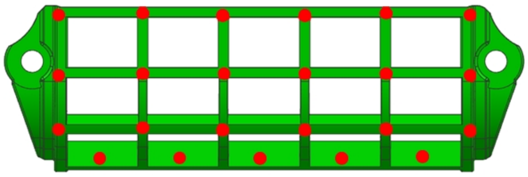
Slides or jaws are required for **molded parts with undercuts**, as straight retraction is not possible here. Undercuts are areas of the molded part that cannot be removed by simply opening the mold in the parting line. In this context, one speaks of secondary demolding directions.

In principle, the use of slides should be avoided, as molded parts that can be demolded by straight retraction are particularly efficient.

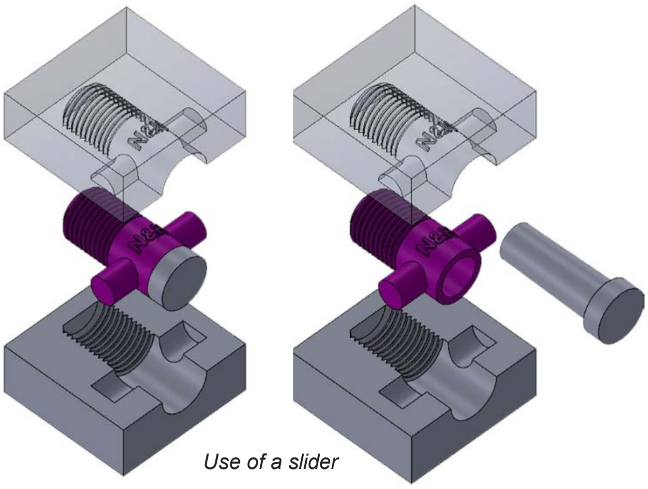
The use of form-fit cores can an effective solution for avoiding sliders. These make it possible to produce the undercut and demold in the main direction through an opening in the component. This technique is often used in the design of snap-fit hooks.

Sometimes it is also possible to avoid undercuts by placing the **parting line** of the molded part so that it crosses the undercut.

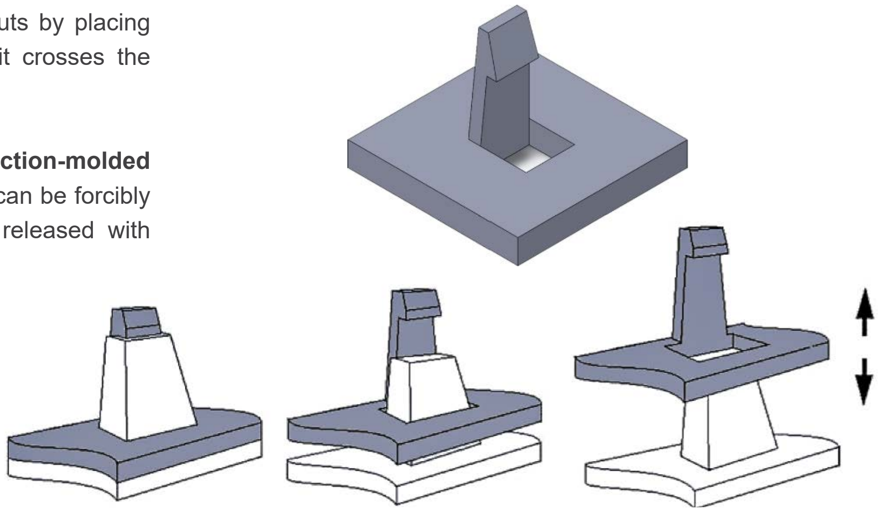
There are several options for demolding **injection-molded parts with internal or external threads**: they can be forcibly demolded, unscrewed with rotating cores or released with collapsible cores.



Positioning of the ejectors (DFM Report)



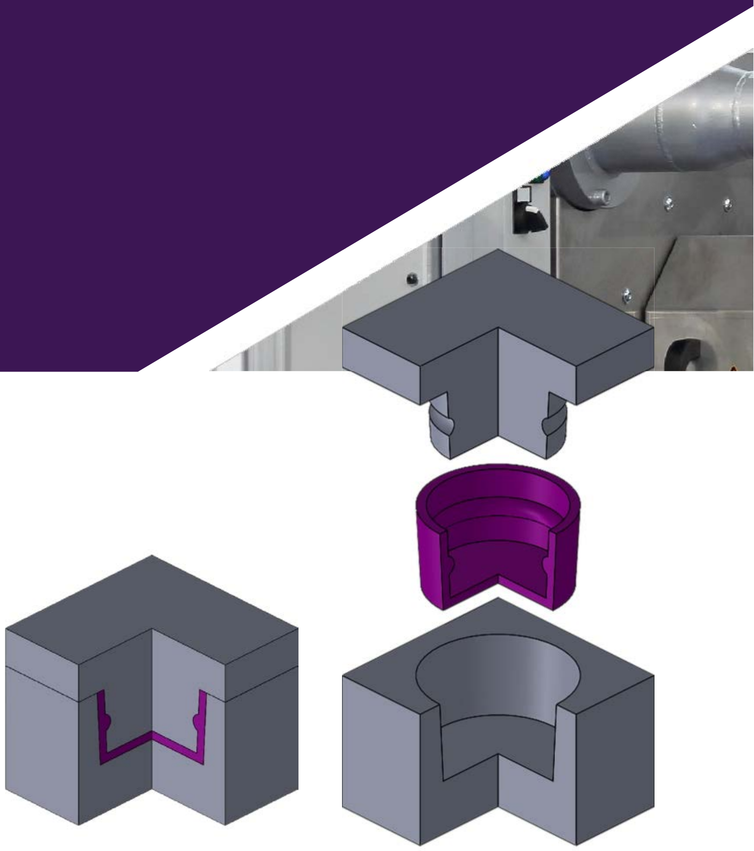
Use of a slider



Solution with form-fit core

PLASTIC MOLDED PARTS

TOOL CONSTRUCTION



SPECIAL CASE ELASTOMER

With elastomer components, undercuts are possible to a certain degree, depending on the material used.

When the component is demolded, the wall of the mold can give way to such an extent that the part is temporarily deformed, but returns almost completely to its original shape after demolding.

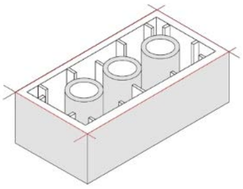
SEPARATING LINE

In injection molds, the parting line is the area where the mold opens and closes. A carefully thought-out parting line design helps to ensure the quality of the finished product, minimize aesthetic defects and optimize production costs.

Positioning the dividing line

Often the parting line runs exactly in the center of the injected part, but this is not always the best practice. Ideally, the parting line should be placed in less visible areas of the part to minimize aesthetic defects such as visible burr lines.

A good example of this is a LEGO® brick. Instead of placing the dividing line in the center of the top of the stone, it runs along the bottom edges. This makes it less conspicuous and improves the visual appearance of the product.

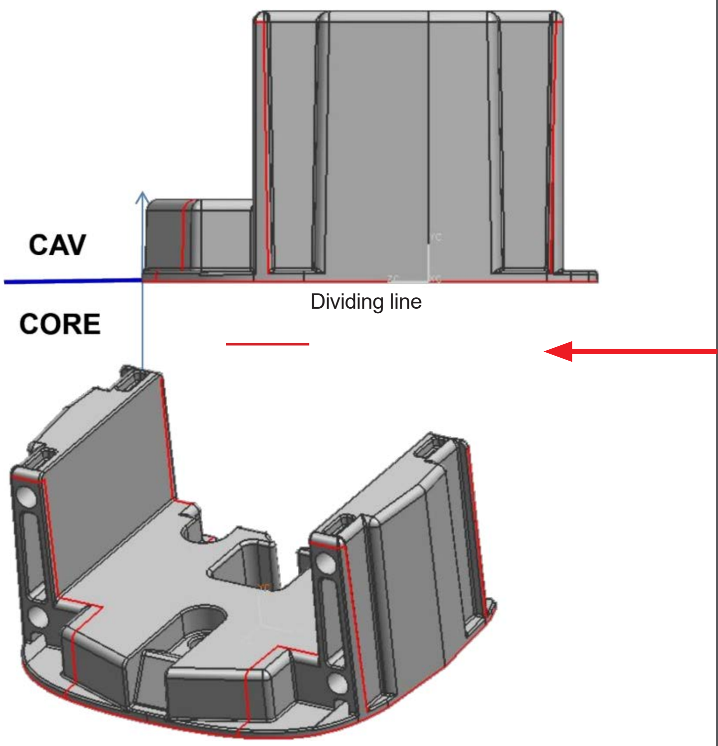


Sharp edges

Sharp edges are preferred positions for parting lines as they simplify the design and machining of the mold. These areas provide clear and defined parting surfaces that minimize flash and therefore reduce rework. also help to manufacturing costs as they allow for simpler mold design.

Rounded surfaces

Rounded (fillet) surfaces, on the other hand, are less ideal for parting lines. They often require a tighter tolerance in mold construction, which increases costs. There is also a higher risk of flash, which occurs when the two halves of the mold do not close perfectly. To avoid these problems, parting lines should preferably not be placed on rounded surfaces.

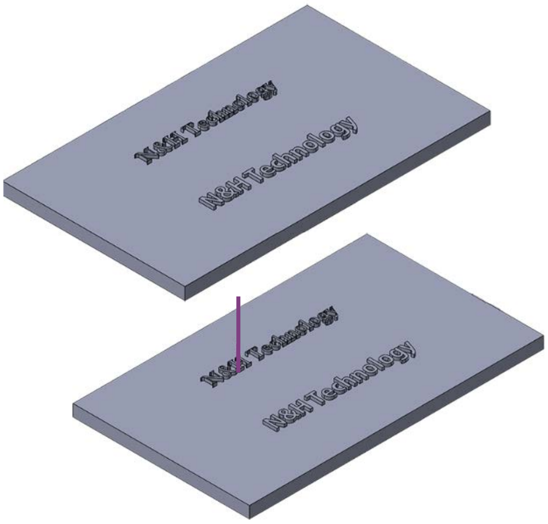


LIMITS OF THE DESIGN

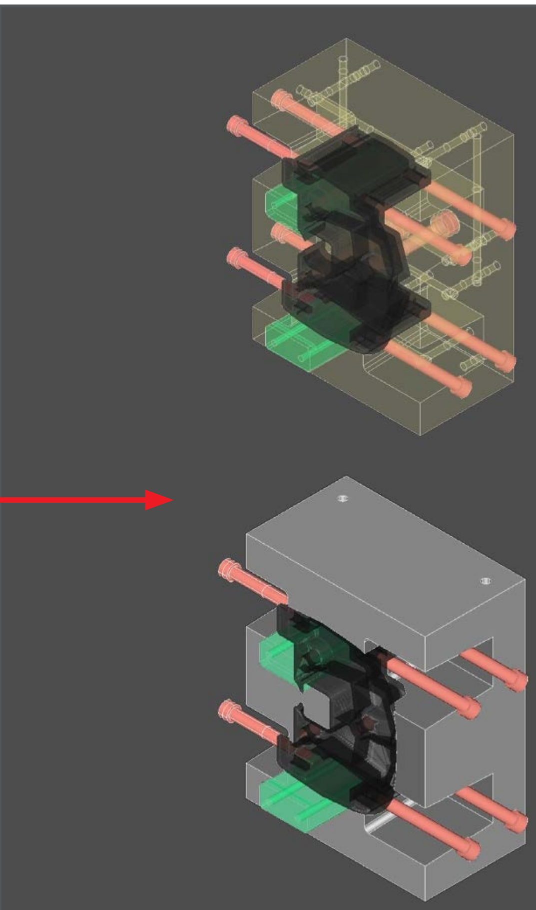
During planning and design, the technical limits must be taken into account from the outset. Not only does the desired plastic have specific parameters, but the production of the tool also has its limits.

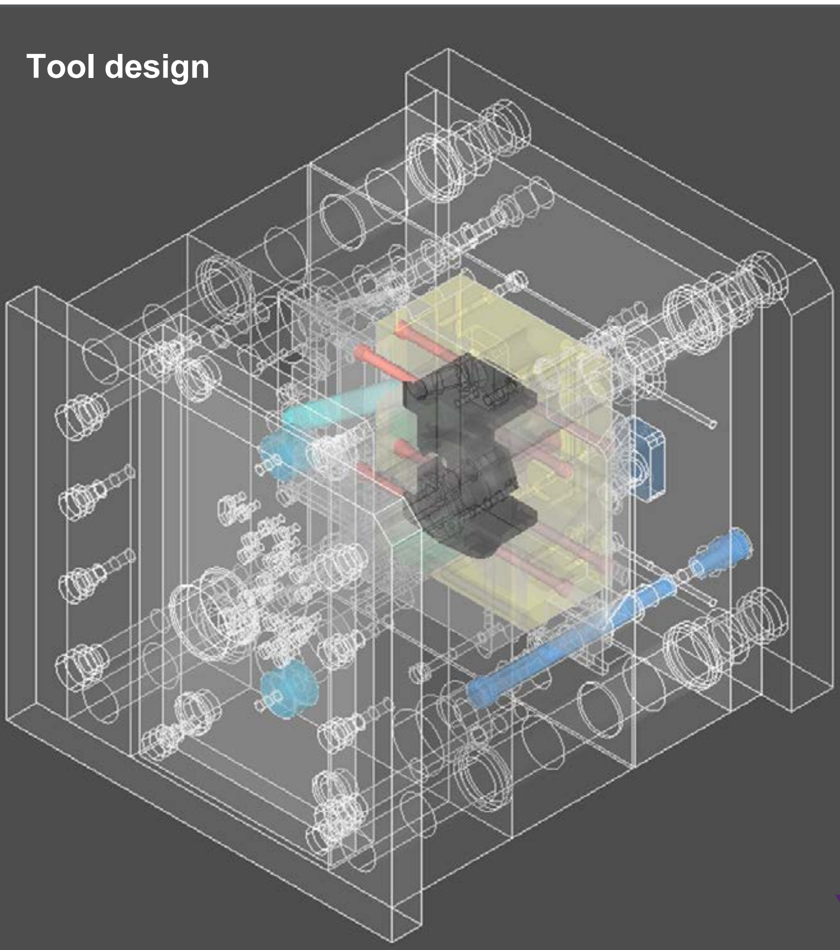
For example, it is possible to incorporate logos or lettering into the mold. However, it must be ensured that the plastic can flow around or fill them completely. Channels that are too narrow can easily remain free of plastic mass.

It is also important to take a realistic view of the possibilities for tool production. Milling heads, such as those used for milling lettering, are always round and therefore generate a certain minimum radius. These radii can usually be designed to be minimal, but must still be taken into account in the design.



Tool design





PLASTIC MOLDED PARTS

APPROACH TYPES

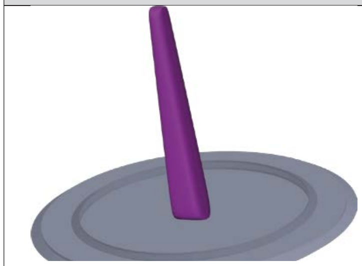
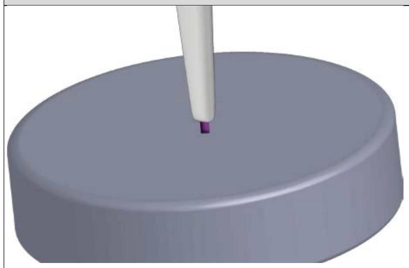
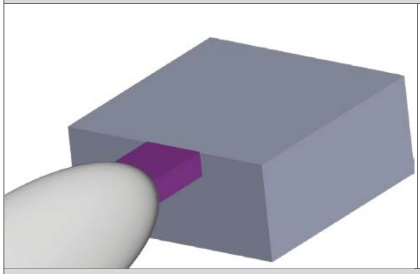
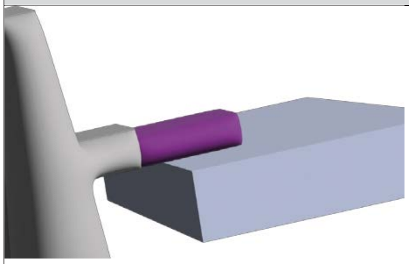
The sprue system guides the molten plastic into the mold. Correctly designed sprues allow the melt to be transferred evenly and quickly so that the cavities can be filled and cooled efficiently.

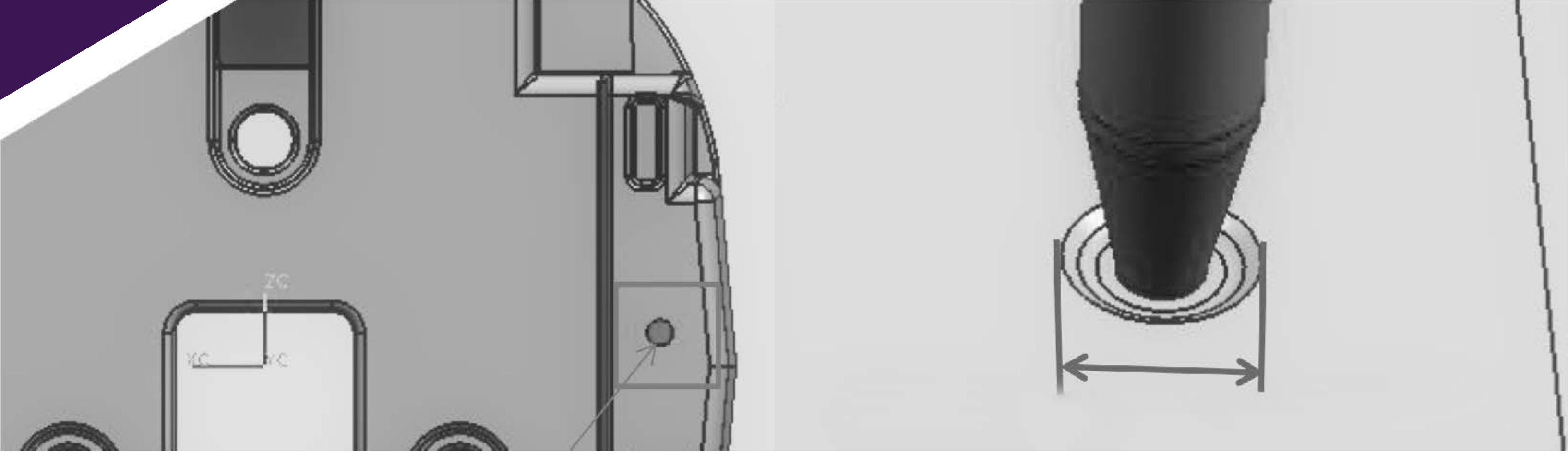
Ideally, the sprues should placed in inconspicuous, non-functional areas and preferably on the thickest part of the molded part. The position is crucial as it influences where deformations, weld lines, sink marks, cavities or other injection molding defects can occur.

At the beginning of the design, it is advisable to use smaller sprues that can be enlarged if necessary.

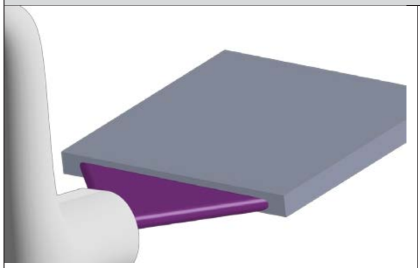
The usual sprue thickness is 50-80% of the thickness of the molded part, with the final diameters of point and tunnel sprues typically between 0.25 and 2.0 mm. Shorter sprues also help to reduce the pressure drop in the sprue area.

The most common gate types are:

Sprue gate / sprue spigot	
	• Simple production, fast injection of large volumes • Ideal for round / cylindrical components • Exact concentration of the material • Frequently used for low quantities and single cavity molds • Visible sprue marking, must be removed or processed manually.
Pin Gate / Point gate	
	• Ideal for 3-plate molds, facilitates symmetrical filling • Use for small and precise components, as well as for thin-walled parts • Suitable for multi-cavity molds • Minimal visible marking on critical surfaces • High precision and surface quality • Diameter: 40 to 50% of the wall thickness, web height: 0.5 to 1.0 mm
Edge gate / corner gate	
	• Used for components with large surfaces and thin-walled parts • Sprue is arranged on the parting line • Fills the mold from the side, top or bottom • Typically rectangular cross-section for an even distribution of the synthetic material. fabric along the edge • Thickness: 50 to 80% of the wall thickness, web height: 0.5 to 1.5 mm
Overlap Gate	
	• Suitable for applications where smooth surfaces are crucial • Helps to avoid the risk of splashes • Reduces internal tension through gradual pressure distribution • Similar to an edge cut, but overlaps the surface • Thickness: 0.4 to 6.4 mm / Width: 1.6 to 12.7 mm

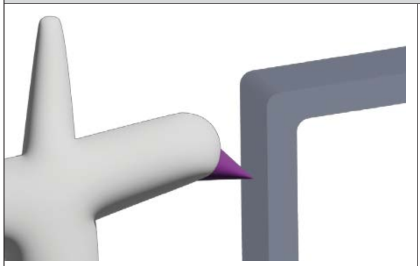


Fan gate / fan gate



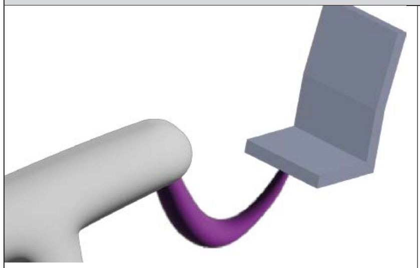
- Gradual widening of the gate to a fan shape in the direction of the mold cavity with uniform thickness
- Often used to ensure a stable flow in large parts
- Uniform filling of the cavity, important for warpage control and dimensional accuracy
- Maximum thickness of the fan gate: 75 % of the part wall thickness
- Typical gate thicknesses: 0.25 to 1.6 mm

Tunnel gate / submarine sprue



- Angled, tapered tunnel that runs from the end of the runner to the cavity
- Diameter of the tunnel: 30 to 70% of the wall thickness of the part
- Hidden bleed that creates a clean surface without visible bleed marks enables
- Efficient separation of the sprue from the finished part, minimizes post-processing

Cashew / Banana Gate



- Curved tunnel gate that runs under the parting line
- Used for components with demanding surface quality, as the chamfer is is hidden and leaves no visible markings
- Ideal for parts with complicated geometries or areas that are difficult to access
- Automated separation of the sprue
- Diameter of the tunnel: typically 0.25 to 2.0 mm

MATERIAL RECOMMENDATION

	PVC	PE	PP	PC	PS	PA	POM	AS	ABS	PMMA	SFT
Sprue	√	√	√	√	√	√	√	√	√	√	√
Pin		√	√	√	√	√	√	√	√		
Edge	√	√	√	√		√	√	√	√	√	√
Overlap							√	√	√	√	
Fan			√				√		√		
Tunnel					√	√	√		√		
Cashew	√	√	√	√	√	√	√	√	√	√	√

2K INJECTION MOLDING

Multi-component injection molding is a manufacturing process that uses two or three different materials in a single injection molding cycle.

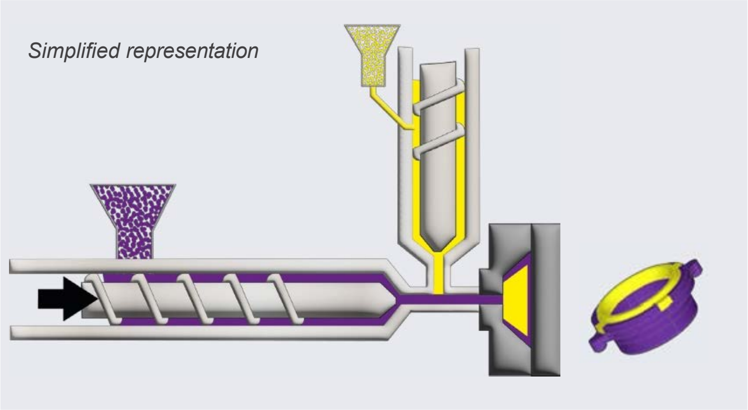
It enables the production of complex or multi-part products in a single step and is particularly useful for products that require a combination of different properties such as hardness, color or material type.

Functionality and machine configuration

In the case of 2 components, 2-component injection moulding requires specialized machines with two separate injection units. Each unit is responsible for injecting a specific material. The machines can be configured to inject the materials into the mold either sequentially or simultaneously. Precise control systems are required to accurately control injection timing, pressure and material volumes.

Tool design and precision

The mold in 2-component injection molding must be specially designed to accommodate two materials. It usually contains multiple cavities and cores that can move or rotate to mold the first material and then be ready for the injection of the second material. This can be done by rotating the mold or by moving cores within the mold.



Material combination

A decisive success factor in the 2K injection molding process is the right combination of materials. Not all thermoplastics adhere optimally to each other or offer the desired mechanical, chemical or thermal properties. Careful selection of the materials is therefore essential to ensure reliable adhesion, high load-bearing capacity and long-term durability of the components.

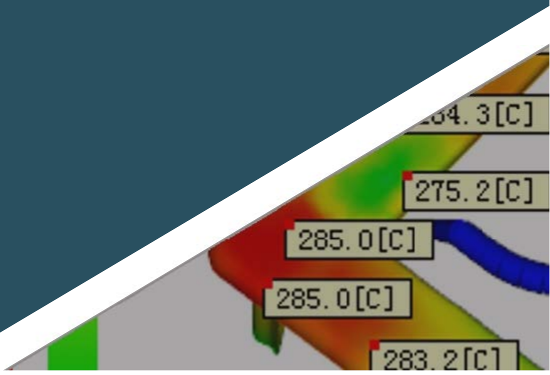
To make it easier for developers and designers to select the ideal material pairings, we have compiled an overview of materials. This shows which plastic combinations can be joined together particularly well and which are less suitable.

🟩 very good
🟧 good
🟥 bad
🟦 not possible

Material combinations		ABS	ASA	CA	EVA	PA 6	PA 6.6	PBT	PC	PE	PET	PMMA	POM	PP	PPO mod.	PS	PSU	SAN	TPE	TPU	EPDM	NR/SBR	SBR	LSR
Thermoplastics	ABS	🟩	🟩	🟩	🟦	🟦	🟦	🟩	🟩	🟦	🟩	🟩	🟦	🟦	🟧	🟧	🟦	🟩	🟥	🟧	🟦	🟦	🟦	🟦
	ABS / PC	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟧	🟦	🟦	🟦	🟦
	ASA	🟩	🟩	🟩	🟩	🟦	🟦	🟦	🟧	🟦	🟦	🟧	🟦	🟦	🟧	🟧	🟦	🟩	🟦	🟩	🟦	🟦	🟦	🟦
	CA	🟩	🟩	🟩	🟥	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟧	🟦	🟩	🟦	🟩	🟦	🟦	🟦	🟦
	EVA	🟦	🟩	🟥	🟩	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦	🟩	🟦	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦
	PA 6	🟦	🟦	🟦	🟦	🟩	🟩	🟦	🟦	🟥	🟦	🟦	🟥	🟦	🟦	🟦	🟦	🟦	🟥	🟧	🟥	🟥	🟥	🟩
	PA 6 (mod. +25% GF)	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟧	🟦	🟦	🟦	🟦
	PA 6.6	🟦	🟦	🟦	🟦	🟧	🟩	🟩	🟧	🟥	🟦	🟦	🟥	🟦	🟦	🟧	🟦	🟦	🟥	🟧	🟥	🟥	🟥	🟩
	PA 6.6 (mod. +25% GF)	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟧	🟦	🟦	🟦	🟦
	PA 6.12	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦
	PA 6.12 (mod. +25% GF)	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟧	🟦	🟦	🟦	🟦
	PBT	🟦	🟦	🟦	🟦	🟦	🟩	🟩	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟦	🟦	🟧	🟧	🟩	🟥	🟥	🟩
	PC	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟧	🟩	🟩	🟧	🟦	🟦	🟦	🟧
	PC / PBT	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟧	🟦	🟦	🟦	🟩
	PE	🟧	🟧	🟧	🟩	🟥	🟥	🟦	🟧	🟧	🟦	🟥	🟥	🟧	🟦	🟧	🟦	🟦	🟥	🟥	🟦	🟦	🟦	🟦
	PET	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟦	🟧	🟦	🟦	🟦	🟦	🟩	🟥	🟥	🟦	🟦	🟦	🟦
	PMMA	🟧	🟧	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟦	🟩	🟦	🟦	🟦	🟥	🟦	🟧	🟦	🟧	🟦	🟦	🟦	🟦
	POM	🟧	🟧	🟦	🟦	🟥	🟥	🟦	🟦	🟥	🟧	🟦	🟩	🟥	🟦	🟧	🟦	🟧	🟥	🟧	🟦	🟦	🟦	🟦
	PP	🟧	🟧	🟧	🟩	🟥	🟥	🟦	🟧	🟧	🟦	🟥	🟥	🟧	🟥	🟧	🟦	🟧	🟥	🟥	🟦	🟦	🟦	🟧
	PPO mod.	🟧	🟧	🟧	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟩	🟩	🟦	🟩	🟥	🟥	🟦	🟦	🟦	🟦
	PPE mod.	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟩	🟩	🟦
	PS	🟧	🟧	🟧	🟩	🟧	🟧	🟦	🟧	🟦	🟦	🟦	🟦	🟦	🟩	🟩	🟦	🟧	🟥	🟥	🟦	🟦	🟦	🟦
	PSU	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟦
	Rigid PVC	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟥	🟩	🟦	🟦	🟦	🟦
	SAN	🟩	🟩	🟩	🟦	🟦	🟦	🟩	🟩	🟦	🟩	🟩	🟧	🟦	🟧	🟧	🟦	🟩	🟥	🟩	🟦	🟦	🟦	🟦
TP	TPE	🟦	🟦	🟦	🟦	🟦	🟦	🟧	🟩	🟦	🟧	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦
	TPU	🟩	🟩	🟩	🟦	🟩	🟩	🟦	🟩	🟦	🟦	🟦	🟦	🟦	🟧	🟧	🟦	🟩	🟦	🟩	🟦	🟦	🟦	🟦
Elastomers	EPDM	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦	🟦
	NR	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦	🟦
	SBR	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩	🟦
	LSR	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟩

TOOL CONSTRUCTION

MOLDFLOW ANALYSIS



Plastics have a specific viscosity that determines their flow behavior. However, the viscosity of plastics is not a constant; it generally decreases as a function of temperature and decreasing strain rate. The terms toughness, ductility and brittleness are used here to evaluate the properties of the materials.

In order to achieve the best possible results, these properties must be cleverly coordinated with the mold design. A key technology for the analysis and optimization of the flow behaviour is the mold flow analysis, which is carried out by our experts.

Moldflow analysis is an advanced simulation method that visualizes the flow of plastic melts during the injection moulding process.

Our experts carry out this analysis in order to precisely predict and optimize the material behaviour in the mould.

ADVANTAGES OF MOLDFLOW ANALYSIS

1. Visualize flow behaviour:

Moldflow provides a detailed visualization of the plastic flow in the mould, allowing bottlenecks and problem areas to be identified.

2. Optimize sprue points:

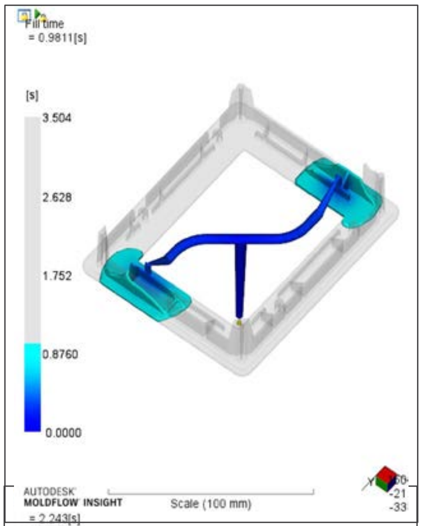
The analysis helps determine the optimal positions for sprue locations to ensure even filling and minimal visible seams or marks.

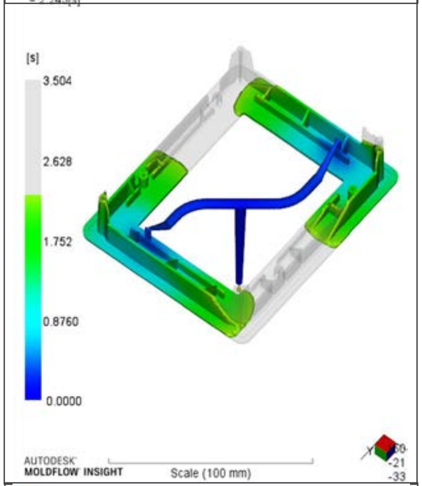
3. Reduce tensions:

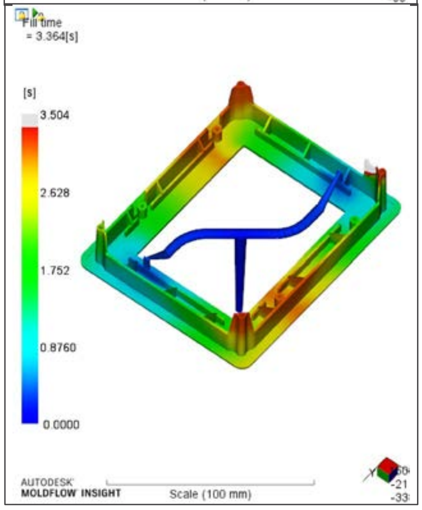
The simulation allows flow-related stresses to be identified and minimized, which improves the mechanical properties and service life of the end product.

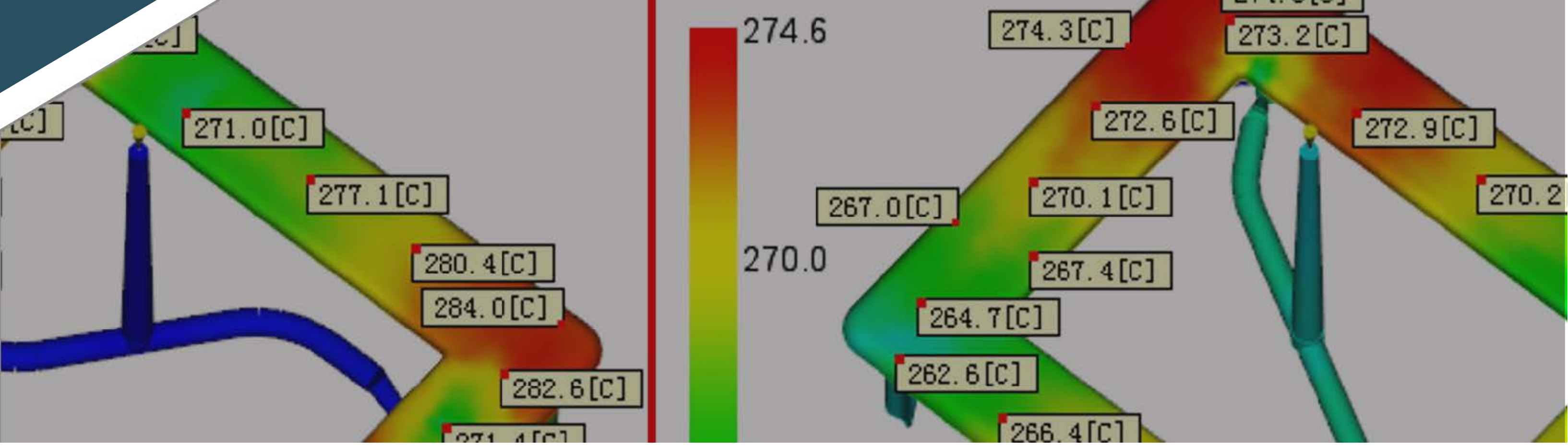
4. Material savings:

Optimized flow paths and injection points reduce material consumption and lower production costs.

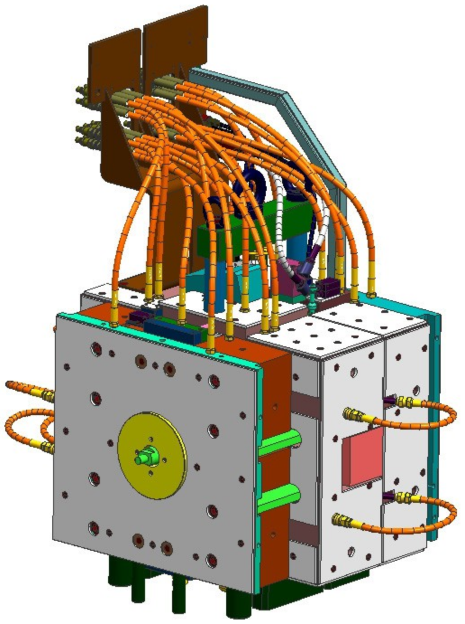








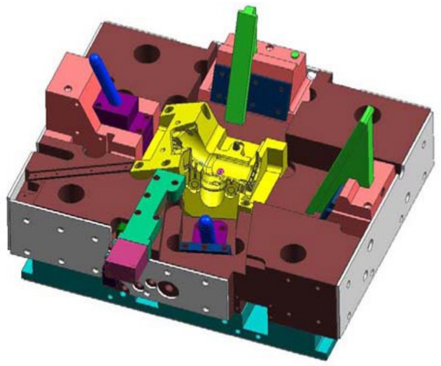
CHALLENGES



Depending on the geometry, complexity and intended material of the moulded part, optimized and individual temperature control and injection systems are already designed during the construction phase of the mould. This ensures that the plastic mass is distributed evenly and that the mold is filled completely and cleanly.

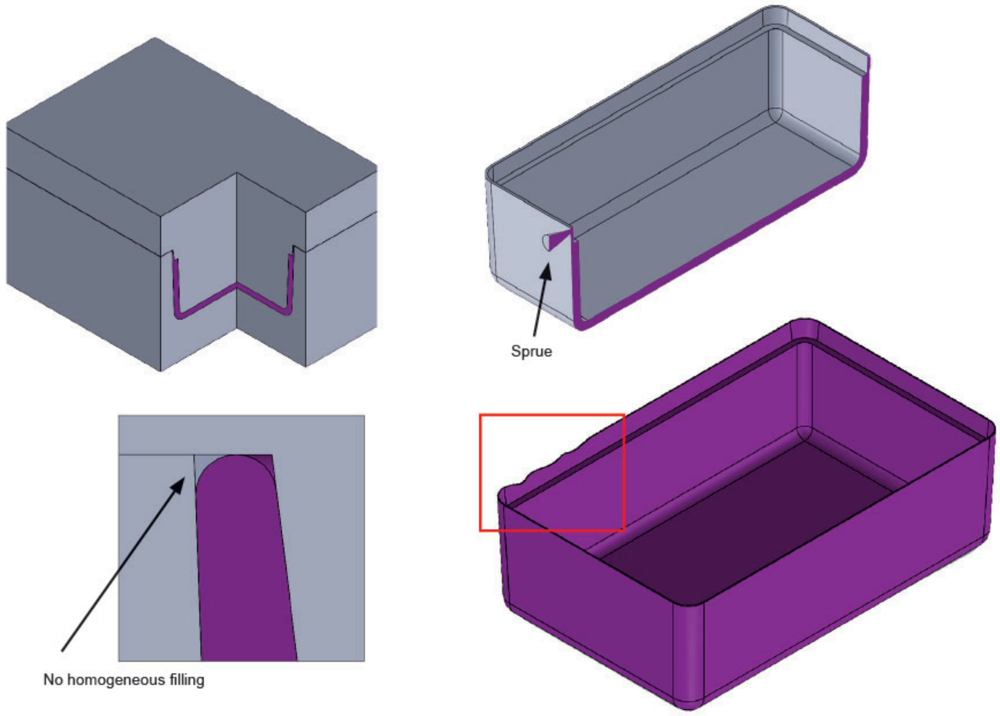
It is important to note that the mold is not a completely closed mold. The plastic material must enter the mold at one or more points. These injection points should be positioned in such a way that they cause as little visual and functional disruption as possible. Our experts will help you to identify the best positions and geometries for the injection points.

OPTIMAL FILLING



An even distribution of the plastic material is essential for a complete and clean filling of the mold.

The suboptimal example shown shows a shape that is difficult to fill completely, which can lead to visual defects and functional limitations. It would be better to fill from the bottom in the middle or in several places at the bottom. This avoids flow-related stresses and ensures uniform filling. **Moldflow analysis** helps to develop such optimal filling strategies.



TOOL FALLING

SURFACES

STRUCTURES

The tool-falling method for surface finishing of plastic parts makes it possible to integrate the desired surface structure directly during the manufacturing process.

This technique allows the texture and **design of the surface to be applied directly to the mold.** The finished part is given the desired surface finish without need for additional post-processing steps such as painting or milling.

The targeted **alternation between textured and polished areas** creates high-contrast and aesthetically pleasing design surfaces. In addition, a surface structure also **conceals** irregularities such as flow seams, weld lines and sink marks that can occur during plastic injection molding.

VDI 3400 and SPI are two established standards for the **surface classification** of injection molds. Both systems provide guidelines for determining and describing surface quality, but with different approaches and focuses.

VDI 3400 is defined by the Society of German Engineers and comprises 45 different **texture classes** with varying degrees of roughness. The classes are numbered (e.g. VDI 12), with lower numbers a smoother surface and higher numbers a coarser texture.

The **SPI standards** divide surfaces into four main categories (A, B, C, D), which in turn are subdivided into specific subcategories (e.g. SPI A-1, A-2, A-3). This classification based on the **surface quality** achieved by different **polishing processes**. A-grade surfaces are high-gloss, while D-grade surfaces are textured, matt.

SURFACE RECOMMENDATION

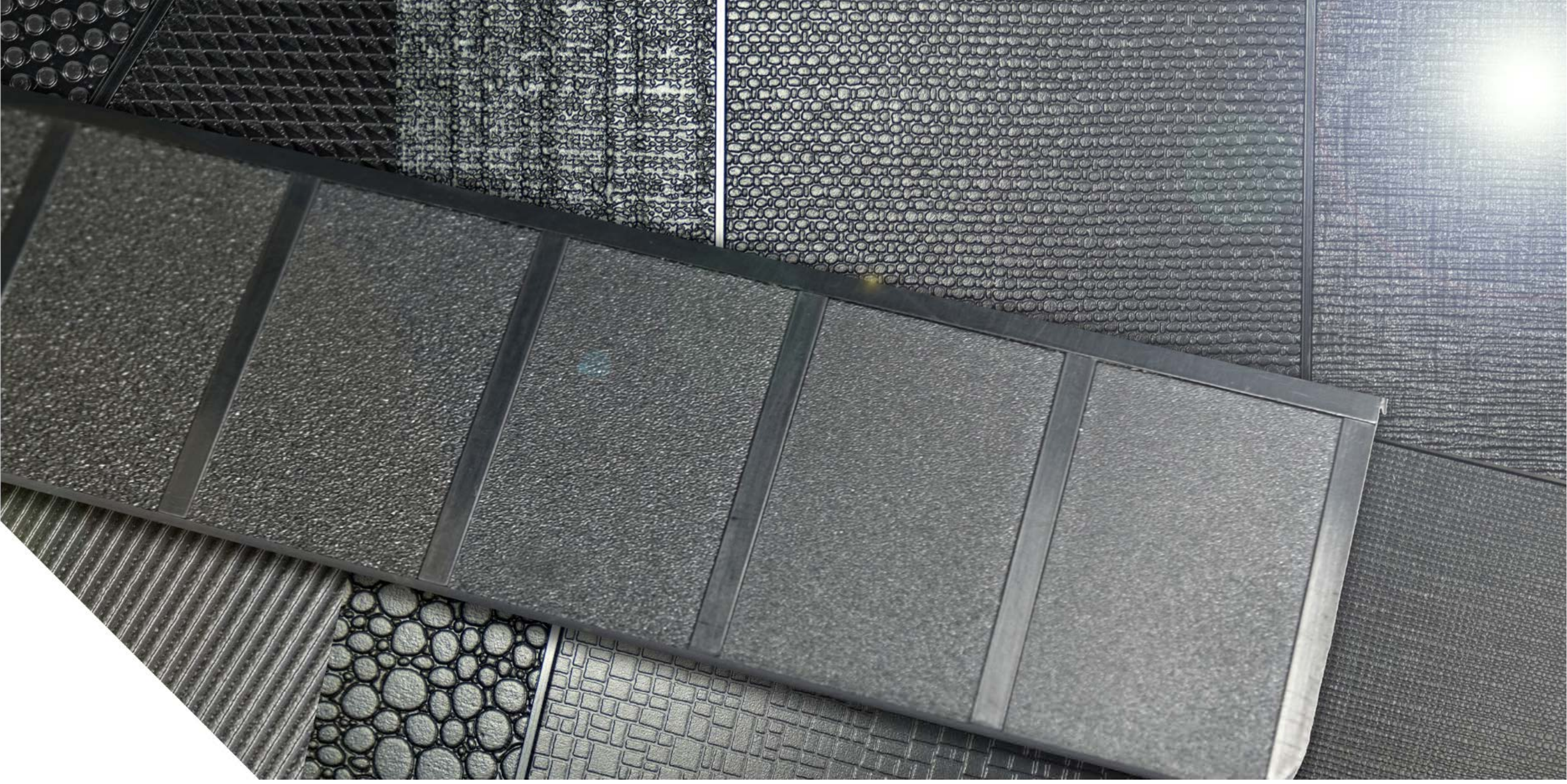
Surface finish			
🟩	very good	🟧	average
🟨	good	🟥	not recommended

	SPI	Ra µm	PC	PS	PP	ABS	TPU
High-gloss	A-1	0.012 - 0.025	🟧	🟧	🟥	🟧	🟥
	A-2	0.025 - 0.05	🟨	🟧	🟧	🟧	🟥
	A-3	0.05 - 0.10	🟩	🟨	🟧	🟨	🟥
Shiny	B-1	0.05 - 0.10	🟨	🟩	🟨	🟨	🟥
	B-2	0.10 - 0.15	🟨	🟩	🟨	🟨	🟧
	B-3	0.28 - 0.32	🟧	🟩	🟩	🟩	🟧
Matt	C-1	0.35 - 0.40	🟧	🟩	🟩	🟩	🟨
	C-2	0.45 - 0.55	🟥	🟩	🟩	🟩	🟨
	C-3	0.63 - 0.70	🟥	🟩	🟩	🟩	🟨
Structure	D-1	0.80 - 1.00	🟩	🟩	🟩	🟩	🟩
	D-2	1.00 - 2.80	🟥	🟩	🟩	🟩	🟩
	D-3	3.20 - 18.0	🟥	🟨	🟩	🟨	🟨

N&H SAMPLE CARD

Our sample card is molded in black ABS plastic, and shows, from polished to rough, twelve different classifications of SPI surface treatments, as well as the twelve most common VDI 3400 surface textures.

You can request the N&H sample card from us free of charge.



DRAFT ANGLES

The demoulding of a moulded part from the injection mould requires considerable force, especially if the demoulding forces are not optimized. To prevent damage to the molded part and extend the service life of the tool, it is important that these forces are kept as low as possible.

Conicity, also known as the **draft angle, plays a crucial role here**. This angle is integrated into the design of the injection molded part to enable easy removal of the finished part from the mold.

The taper not only helps to reduce demolding forces, but also protects the surface structure of the mold.

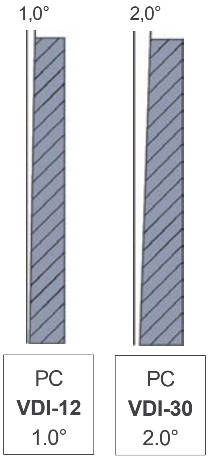
With textured surfaces in particular, it is important to maintain a sufficient bevel to prevent damage to the texture during demolding.

In addition, all sealing surfaces in the mold must a taper. Without this taper, vertical surfaces cannot be aligned correctly, which leads to burr formation on the molded part.

The table shows the recommended minimum draft angles for the common VDI surface structures, based on a wall thickness of 2 mm. A larger bevel should be selected for glass fiber reinforced and filled thermoplastics.

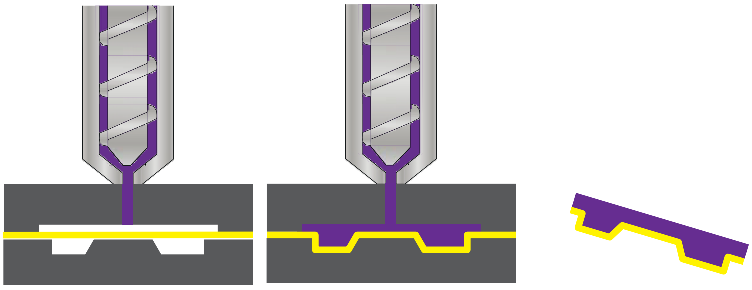
VDI-No.	Ra µm	~ Rz µm	Demolding slope °		
			PA	PC	ABS
12	0,40	1,5	0,5	**1,0**	0,5
15	0,56	2,4	0,5	1,0	0,5
18	0,80	3,3	0,5	1,0	0,5
21	1,12	4,7	0,5	1,0	0,5
24	1,60	6,5	0,5	1,5	1,0
27	2,24	10,5	1,0	2,0	1,5
30	3,15	12,5	1,5	**2,0**	2,0
33	4,50	17,5	2,0	3,0	2,5
36	6,30	24,0	2,5	4,0	3,0
39	9,00	34,0	3,0	5,0	4,0
42	12,50	48,0	4,0	6,0	5,0
45	18,00	69,0	5,0	7,0	6,0

Ra: Surface roughness
Rz: Level of roughness



SURFACES

IN-MOLD DECORATION (IMD)



In-Mould Decoration (IMD) enables the integration of decorative surfaces directly into the plastic component.

The designs are clear, detailed and resistant to wear and tear, fading, mechanical stress and environmental influences.

Functionality

The prepared design film is precisely inserted into the injection mold. The pressure and heat during the injection molding process bonds the film to the plastic, permanently integrating the design into the component. The film fuses with the plastic and forms a robust surface.

After cooling, the finished plastic part is removed from the mold. The decorative design is now an integral part of the component.

Cost efficiency

By integrating the decoration into the injection molding process, additional steps and costs for subsequent surface finishing are eliminated. This can lead to more efficient and cost-effective production.

FURTHER PROCEDURES

By using various techniques such as painting, printing or coating, plastic surfaces can be not only more resistant to abrasion, but also more visually appealing.

These processes extend the service life of plastic parts and improve their application possibilities. They also enable individual surface design, which can be of great importance for brand identity and product individualization.

PRINTING

Moulded plastic parts are printed using various printing techniques such as pad printing, screen printing and digital printing. Printing makes it possible to customize the components with logos, patterns or technical information.

Durability and longevity

The inks and colors used are often UV-resistant and abrasion-resistant, which increases the longevity of the product. This makes printed plastic parts particularly resistant to external influences.

Efficiency and cost-effectiveness

Printing technology enables fast and cost-effective production of components with complex and colorful designs without the need extensive pre- or post-processing. This leads to efficient and economical production that meets the requirements of modern industry.

Preferred color specifications: RAL, Pantone or color sample.

TAMPON PRINTING

Pad printing is particularly suitable for printing on irregular or curved surfaces. This technique is often used for small to medium-sized plastic parts and enables precise and detailed designs, even on complex geometries.

SCREEN PRINTING

Screen printing is ideal for flat or slightly curved surfaces and large print areas. This method is often used for large decorations on plastic parts. It offers high color density and is cost-effective for large print runs.

DIGITAL PRINT

Digital printing is perfect for smaller print runs or individual, customized designs. This technique offers maximum flexibility in color design and can react quickly to changing designs. Digital printing is often used for prototypes, personalized products and complex graphics.

PLASTIC MOLDED PARTS

SURFACE FINISHING

PAINTING

Painting not only enhances the appearance of plastic parts, but also increases their durability and resistance.

Aesthetic enhancement

The targeted application of layers of lacquer gives the parts an even, smooth surface that is both visually appealing and pleasant to the touch.

Modern painting techniques offer a wide range of colors and effects, from high-gloss and metallic finishes to matt and textured surfaces, allowing for enormous design flexibility.

Protection from environmental influences

A painted plastic part is better protected against environmental influences. The coating acts as a barrier against UV radiation, moisture and chemical corrosion. This considerably extends the service life of the plastic parts.

Improvement of mechanical properties

A painted surface is harder and more resistant to abrasion and scratches. This is particularly advantageous for components that are used in demanding environments and are exposed to high mechanical loads.

SOFT-TOUCH LACQUERING

The soft-touch finish gives the plastic parts a pleasant, velvety surface that is not only visually appealing, but also conveys a high-quality, soft feel.

Soft-touch coating is often used in areas where the feel of the product plays an important role, such as automotive interior trim, electronic housings and consumer goods.

LEAD LACK

A conductive coating gives the plastic parts conductive properties that make them ideal for protection against electromagnetic interference and electrostatic discharge.

Conductive lacquer contains conductive particles such as carbon, silver or copper. This coating is particularly suitable for applications that require high electromagnetic compatibility (EMC) and ESD protection, such as electronic housings.

LASER ENGRAVING / TEXTURING

Laser marking provides a precise and efficient method of marking plastic parts with various information such as logos, serial numbers, barcodes or decorative designs. Additionally, laser texturing can be used to create detailed patterns and textures on the surface of plastic parts, improving both functionality and aesthetics.

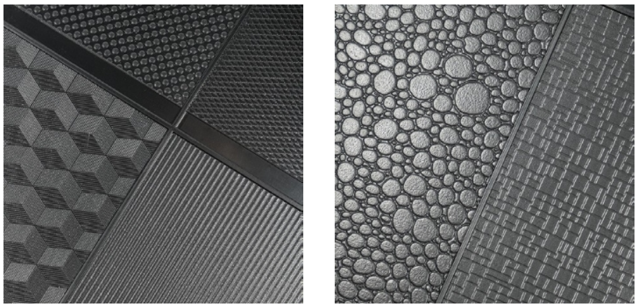
LASER MARKING

The laser beam removes a thin layer of the material and exposes the layers underneath. The color of the underlying layer determines the legend color. If the base color is omitted, the color of the plastic material itself determines the legend color.

LASER TEXTURING

The laser beam can also be used to create textures by selectively removing material or changing the surface properties.

This process enables the creation of complex patterns, microtextures and surface modifications that can improve grip, reduce friction or provide unique visual and tactile effects.



PVD COATING

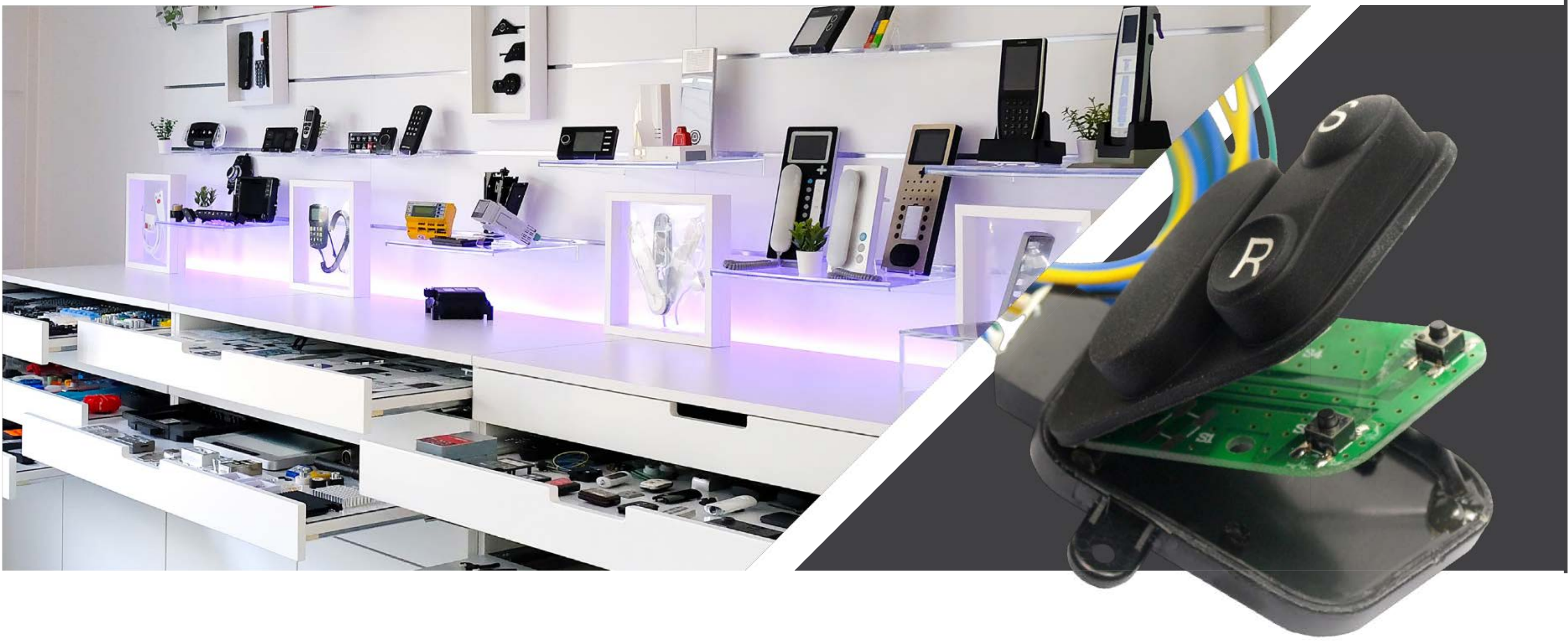
The PVD (Physical Vapor Deposition) coating gives the plastic parts a metallic surface that is both visually appealing and functional.

This coating offers excellent hardness and wear resistance as well as high resistance to corrosion and chemical influences.

PVD coating is often used in areas where both the aesthetics and durability of the product are of crucial importance.

Typical areas of application are automotive interior and exterior parts, electronic devices and consumer goods where a high-quality, metallic appearance is required.

YOUR PROJECT



We are your reliable partner in every phase of your project - from the initial design to series production.

Our aim is to provide you with comprehensive support and ensure the success of your project.

Your request

We would be happy to provide you with a non-binding quote tailored to your individual project. We need the following information:

- Technical drawings, sketches or samples
- Technical specifications
- Details of the desired equipment extras
- Required quantity, annual requirement or term

As soon as we have received this information, one of our experienced engineers will contact you as soon as possible.

To protect your sensitive data, it goes without saying that we sign a non-disclosure agreement (NDA).

Personal advice and meetings

We are always available for technical advice - by phone, online in person by appointment. A face-to-face meeting is often particularly valuable for complex or new projects in order to precisely understand your requirements and needs and offer you the best solution.

Our showroom

In our showroom, you also have the opportunity to see the quality and functionality of our products for yourself, receive individual advice and try out the various models.

Together to success

We look forward to supporting you in the implementation of your projects - let's be successful together!

N&H Technology GmbH

MORE THAN PLASTIC

Your full-service supplier for HMI operating units

We design and manufacture all components and molded parts for individual input units. Do you need a customized membrane keypad or silicone keypad, an injection-moulded housing or cable assembly?

Get in touch with us!

As a full-service supplier for HMI operating units, we develop and manufacture customized input solutions - from the initial idea to series production, precisely tailored to your technical and industry-specific requirements.

SERVICE

Our catalogs offer technical details and design information for optimal product selection - as a print version on request or conveniently available as Download at www.nh-technology.de

N&H Technology GmbH

FOLIENTASTATUREN

N&H Technology GmbH

KUNSTSTOFF - FORMTEILE

N&H Technology GmbH

SILIKONSCHALTMATTEN

N&H Technology GmbH

STANDARD TASTATUREN

N&H Technology GmbH

FEDERKONTAKTE

N&H Technology GmbH

N&H Technology GmbH
Gießerallee 21
D-47877 Willich

T. +49 (0)2154 - 8125 0
F. +49 (0)2154 - 8125 22

info@nh-technology.de
www.nh-technology.de

TÜVRheinland
ZERTIFIZIERT
Management
System
ISO 9001:2015
www.tuv.com
ID 9105016435

Follow us