Guidelines for the Injection Molding of Sarlink

Contents

sarlink

Introduction

Sarlink® thermoplastic vulcanizates (TPV) are based on dynamically vulcanized EPDM rubber particles dispersed in a polypropylene matrix.

Because of this morphology, Sarlink thermoplastic vulcanizates combine the performance characteristics of widely used thermoset rubbers, such as EPDM and polychloroprene, with the ease of plastic processing.

Sarlink offers a wide range of properties valuable to the end user. For instance:

- resistance to high temperature
- resistance to low temperature
- excellent flexural fatigue endurance
- high impact strength
- excellent resistance to chemicals and solvents
- excellent resistance to weathering
- good electrical properties
- high tear strength
- low tension and compression set
- good resistance to oils (particularly harder grades)

Besides their superior performance characteristics, Sarlink thermoplastic vulcanizates, which are fully compounded, ready-to-use pellets, offer the following processing advantages compared to thermoset rubber:

- no compounding
- no vulcanization
- low capital investment
- easy processing on standard thermoplastic equipment for extrusion, injection molding and blow molding
- fast processing (short cycle times)
- thermal stability/wide processing window
- low energy consumption
- recycling of scrap generated during processing
- recycling of parts after service life
- excellent control of product quality and dimensional tolerances
- in-line colorability
- easy weldability (profiles, sheeting...)
- material combinations by co-extrusion, co-injection or multi-layer blow molding to produce hard/soft articles, solid/sponge profiles, et al.

Special grades

In addition to standard types, a wide range of specialty grades are available, including:

- high flow: For thin long parts a special high flow series is available. These grades have an average spiral flow length of 30 % longer in comparison with standard Sarlink thermoplastic vulcanizates
- UV stabilized grades

Because of its combination of excellent finished part properties and easy processing, Sarlink thermoplastic vulcanizates have found many applications in a wide range of markets, including: automotive, building and construction, electrical, mechanical rubber goods, medical and leisure.

Sarlink thermoplastic vulcanizates are widely used in many injection molded parts such as seals, connectors, corner molds, (soft touch) grips, (car) mats, gaskets, O-rings, body plugs, bumpers, dampers, grommets, protectors, fittings, wheels and many other applications in all market segments.

This brochure is a guideline for the injection molding of Sarlink. Furthermore this brochure contains information about multi-shot injection molding, mold design and hot runner systems. A checklist for trouble shooting is available on the last pages.

® Registered trademark of DSM

sarlink

Characteristics of the Typical Injection Molding Equipment for Sarlink

Sarlink can be easily processed on standard polyolefin thermoplastic injection molding equipment without further modification.

To obtain the optimum injection molding performance, or if dedicated equipment for processing Sarlink has to be purchased, it is recommended to take following items into consideration:

- conventional thermoplastic injection molding equipment
- clamping force: 0.5 to 0.9 Mton/cm2 of projected shot area, including gates, runners, etc
- screw:
  - general purpose polyolefin type
  - L/D ratio 20:1 to 25:1
  - compression ratio of at least 2.5:1
  - compression ratio of 3:1 is optimum, 2.5:1 is minimum 3 barrel capacity: ideally between 2 and 4 weight shots
- temperature control capable of maintenance 280 °C. (Digital PID controllers are recommended)
- injection nozzle: standard type. A hole diameter: 1.5 to 6 mm depending on shot size used is typical
- in some cases the use of a shut off nozzle will improve the surface appearance at the gate
- fume ventilation at the nozzle.

Calculation of machine size

How to calculate the minimal machine size for a given part is shown below:

Calculation of the minimal required clamping force

The most accurate calculation of clamp tonnage required is derived using injection molding flow simulation programs.
These programs calculate press tonnage requirements directly from the filling analysis based on the pressure required to fill the cavity.

In cases where a computer flow simulation is not used, the value can be calculated as follows:

- evaluate the total projected surface area of the part
- multiply this value by the number of cavities. Include cold runners if present
- multiply by the average cavity pressure:
  - 50 N/mm2 for thicker parts (above 1.5 mm)
  - 40 N/mm2 for thinner parts (1.5 mm or less)

An estimation for minimal required clamping force can be made by using the following formula:

$$Fc = Pc\,(Ac \times n + Ar)$$

Fc = clamping force (N)
Pc = average cavity pressure (N/mm2)
Ac = projected surface area of the product (mm2)
Ar = projected surface area of the runner system (mm2)
n = number of cavities.

An inaccurate clamping force can lead to serious production problems. In particular, if the clamping force is too low the part will show flash.

Evaluation of the barrel size

Barrel size is also best derived from flow simulation programs. In cases where a computer flow simulation is not used the value can be calculated as follows:

- calculate the total shot weight including all cavities and runners
- the barrel size should be 2 to 4 times the shot weight

If the barrel capacity is too small or too large, constancy problems in processing will occur.

When choosing equipment for Sarlink, it should be remembered that large injection molding machines tend to have less precision than smaller ones.

Processing Conditions for Optimal Injection Molding of Sarlink

General principles

- The viscosity of Sarlink thermoplastic vulcanizates is more sensitive to shear than to heat. Figures 1 and 2 show these influences.
- Optimum moldings are usually obtained with a relatively high injection speed.
- The material melt temperature has influence on cycle time, surface appearance and weld-line strength.

* Spiral thickness 1.6 mm, width 10 mm.

Drying Sarlink

Sarlink should be processed with a moistures content of less than .06%. Although Sarlink is dried prior to packaging, it can pick up some moisture during storage, especially when packaged in gaylords.

Recommended drying conditions:

- dry minimal 3-4 hours at 80 °C (or 3 hours at 70 °C for softer grades) in a desiccant dryer with a dew point of -40°C.

Recommended storage conditions

The storage conditions for Sarlink are important because of the fact that Sarlink granules will pick up moisture to a slight degree.

Suggested storage conditions for Sarlink are:

- store Sarlink bags closed and undamaged in a non-humid environment
- open bags just before use 3 close the bags securely when the whole content has not been used completely
- bring cold granules to ambient temperature in the processing room while keeping the bags closed.

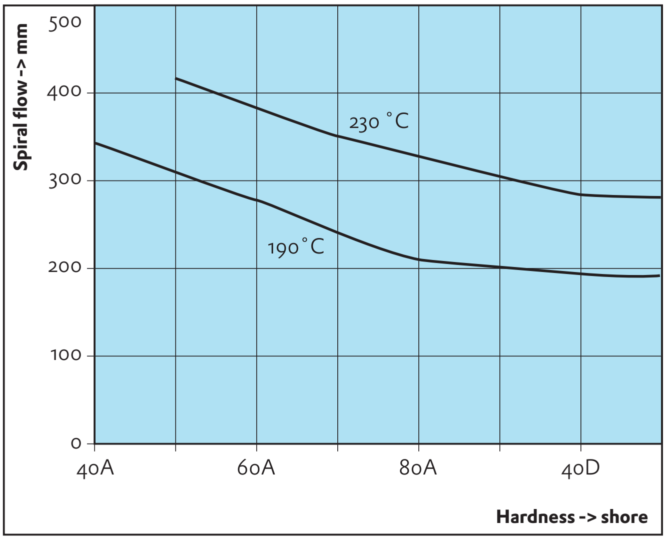


Fig. 1 - The influence of the melt temperature on the spiral flow length* of Sarlink, at a constant injection time of 1 s

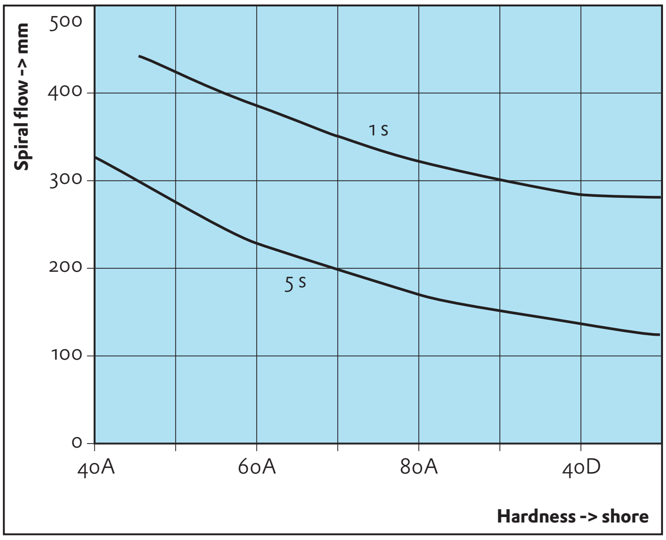


Fig. 2 - The influence of the injection time on the spiral flow length* of Sarlink, at a constant temperature of 230 °C

Use of regrind

Sarlink has excellent melt stability. All scrap generated during processing can be recycled provided it is kept clean.

As an example, figure 3 shows that there is very little variation in physical properties of Sarlink after reprocessing the same material 10 times.

As shown in figure 3, the viscosity of Sarlink slightly decreases after reprocessing. For this reason, it is recommended during production to blend a constant level of less than 20% regrind within the virgin material in order to keep processing conditions constant. Addition of regrind has no negative influence on the surface quality.

As mentioned in the paragraph on drying, it is recommended to dry recycled material before reuse.

Coloring of Sarlink

Natural color Sarlink can be colored by addition of a suitable color masterbatch. Polypropylene based masterbatches offer the best compatibility with Sarlink, but may slightly increase the hardness. Low density polyethylene based master batches are also suitable.

In very critical cases pre-compounded Sarlink based color masterbatches can be used.

Recommended level of color masterbatch in Sarlink is 1 to 5 % by weight to minimize the impact on Sarlink properties.

During injection molding the dispersion of the color masterbatch can be improved if needed, by:

- increasing the back pressure. Beware that this will increase melt temperature and may increase the cycle time.
- decrease the screw speed
- in critical cases, use mixing elements on the screw or a static mixer in the nozzle.

Maintenance of the equipment

Purging of equipment before processing Sarlink

To avoid contamination, the equipment should be cleaned before processing Sarlink. Polyethylene or polypropylene of low melt index is recommended. Purge at a temperature of 200 °C. This cleaning operation is particularly important if the equipment has been used previously for PVC or POM/acetals.

Equipment shut down and cleaning

At the end of the production, run the hopper and screw empty of Sarlink. Purge the barrel thoroughly with polypropylene or polyethylene if change over to another material is anticipated. The use of high viscosity polyolefins will speed up the purging process.
Perform a general clean up of the hopper and any pneumatic conveying lines to remove eventual Sarlink fines. Clean the mold and protect the mold for corrosion.

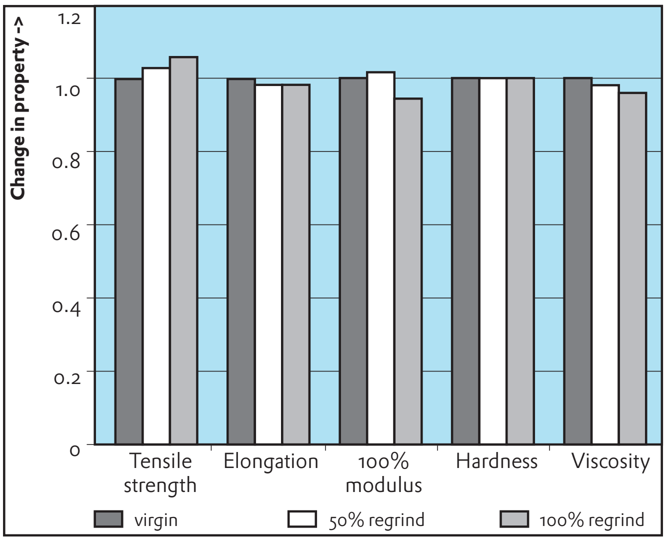


Fig. 3 - Regrind stability of Sarlink

Machinery Settings for the Injection Molding of Sarlink

Temperature settings

Typical temperature settings for the injection molding of Sarlink are shown in table 1.

Processing temperatures (°C)	
Rear	165 - 175
Middle	180 - 200
Front	200 - 220

Table 1 - Typical (average) temperature settings for the processing of Sarlink thermoplastic vulcanisates

Material melt temperature
The material melt temperature for Sarlink thermoplastic vulcanizates typically ranges from 193 °C to 232 °C.

High melt temperatures give:
- better surface appearance
- better adhesion in weld lines
- better adhesion to inserts 3 reduce internal stress
- increased flow length (see figure 1)
- decreased anisotropy
- longer cooling times.

For standard Sarlink grades, it is in general not necessary to have a melt temperature above 250 °C.

Mold temperatures
The chosen mold temperature is a balance between part quality and productivity. A typical mold temperature for the injection molding of Sarlink is 26 °C to 62 °C.

Higher mold temperatures are advisable when:
- parts have a long flow path and/or thin wall sections
- a minimum of warpage is required 3 a good surface appearance is required.

High mold temperatures will, of course, increase cooling times.

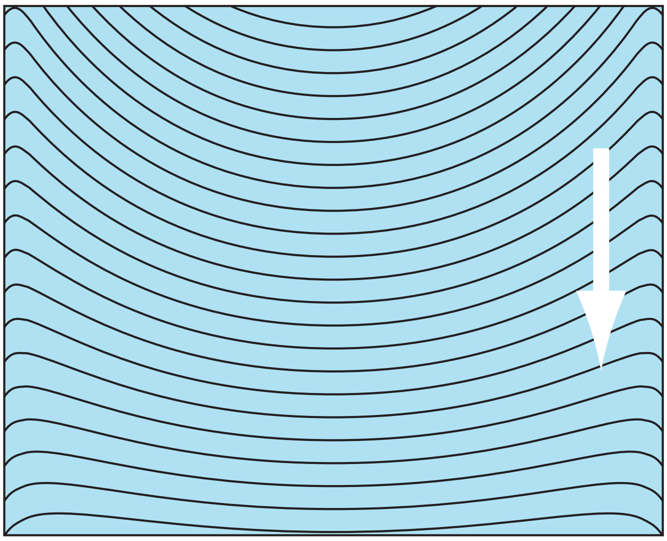


Fig. 4 - Example of 'tiger stripes'; -> injection direction

(Hydraulic) pressure settings

Back pressure
Typical back pressures for Sarlink are 5 to 20 bar.

- Low back pressure will result in plasticizing problems and an inhomogeneous melt.
- Higher back pressures will improve the dispersion of color master-batches or other added additives. It also will improve the melt homogeneity. The plasticizing time will be longer.

Injection pressure
The injection pressure should be high enough to achieve the desired injection speed. A typical injection pressure for Sarlink is from 60 to 120 bar.

- Higher injection pressures will improve the weld line strength.
- Too high injection pressures may lead to flashing.

Holding pressure
A typical holding pressure for Sarlink is 50 to 70 % of the injection pressure.

- A high holding pressure will improve the adhesion in weld lines and the adhesion to inserts or other materials in a multi-shot process.
- Low holding pressure may lead to sink marks and will increase shrinkage.
- Longer holding times tend to lower shrinkage.
- Excessively high holding pressures can lead to deformations.
- Adjust holding pressures to eliminate flash.

Speed settings

Injection speed
A high injection speed will improve the flow length.

High injection speeds will reduce also the visibility of the so called 'tiger stripes' (see figure 4).

Screw speed
A screw speed of 50 to 100 rpm is typically used.

A lower screw speed will improve dispersion of additives like color masterbatch and will help to maintain low melt temperatures when molding thick parts.

Time settings

The major components of the cycle time are:

Mold filling time
This time is a function of the volume of the part, the material grade, location and number of gates, and the injection pressure. A typical injection time for Sarlink is 0.5 to 2 s.

Holding time
This time should be as low as possible. Thick parts require longer holding times (5 to 10 s). For thin parts shorter holding times are possible (1 to 3 s).

Cooling time
The cooling time depends on:
- the wall thickness of the part
- the melt temperature of the material
- the hardness of the particular Sarlink grade
- the complexity of the part.

Ejection/part removal time
The ejection time is dependent on the hardness of the grade, the complexity of the part and the ejector design. Softer parts need slower ejection. Blade ejectors, which have a larger ejecting surface, can reduce the time before ejection. Air assisted part ejection is often used to eject parts with undercuts. Poppet valves are used in the mold cavity to control the air ejection.

Not included here is the time for manual or robotic part removal.

Cushion (buffer)

A cushion of material should remain in the barrel, in front of the screw, after the injection is completed and the mold is filled. The roles of this cushion are:
- to ensure that there is enough material left for packing the mold
- to prevent metal to metal contact during the injection cycle 3 to minimize wear on the nozzle, screw and barrel.

Special recommendations for the injection molding of thick parts

The following factors will reduce cycle times for thick parts:
- reduce barrel temperatures
- reduce screw speed to a minimum in order to reduce melt temperatures
- use a cold mold, chilled if possible
- use a moderate injection speed to reduce shear heating during filling of the mold.

These recommendations can have negative effects on the surface appearance.

Special recommendations for the injection molding of thin parts

- Use high melt temperatures (200 °C minimum).
- Use fast injection speed.
- Use a high mold temperature (50 to 70 °C).
- Avoid overpacking. Fill the part up to 98 % using first stage injection and keep the holding pressure 1 to 3 s at 50 to 70 % of the injection pressure.

Summarized recommended procedure for mould filling:
- **Fill the mould quickly in 0.5 to 2 s, up to 98% using first stage injection**
- **Keep the holding pressure during 1 to 3 s at 50 to 70% of the injection pressure**

A summary of recommended settings for optimal injection molding of standard Sarlink parts is shown in appendix 3.

Mold Design for Sarlink Injection Molded Parts

Mold balance

It is important to have a well balanced mold, in which all cavities are filled in the same time and under the same pressure. If a mold is not correctly balanced, warpage and distortion will occur, and a part of the production will have to be recycled. Shown are two sets of examples of good and bad mold designs are explained.

Example 1
16 cavity mold
Figure 5 shows a correct design, in which all cavities are at equal distance from the sprue. In this design, all cavities will fill in the same time and with the same pressure.
In contrast, figure 6 exhibits an inadequate mold design, in which some cavities are closer to the sprue. These cavities will be filled first and may be over-packed. On the contrary, cavities further away from the sprue may not fill or fill incompletely. Such a mold design should be avoided for Sarlink.

Example 2
4 cavity mold with unequal parts (family mold)
It is possible to produce parts of different sizes in one mold, provided the runner system is properly flow balanced in order to compensate for the difference in part weight and cavity size.

Figure 7 shows the correct mold design. The runners going to the smaller cavities are longer and/or thinner, so that these smaller cavities fill in the same time and under the same pressure as the bigger ones.

Figure 8 shows an inadequate design. In this design, the runner system is symmetrical; therefore the smaller cavities will fill much more quickly and will be overpacked, while the bigger cavities may not fill completely.

Mold filling computer simulations can be used to determine the proper runner and gate dimensions.

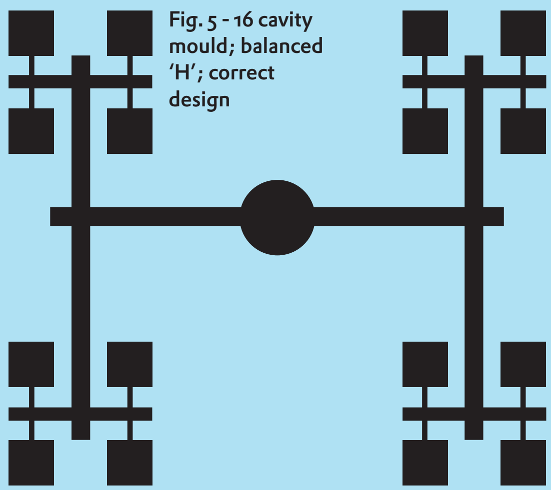


Fig. 5 - 16 cavity mould; balanced 'H'; correct design

Fig. 6 - 16 cavity mould, unbalanced runner; inadequate design

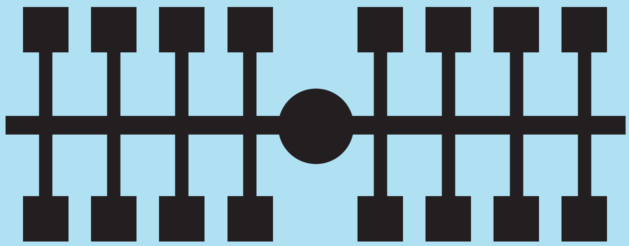


Fig. 7 - Balanced family mould; Correct design

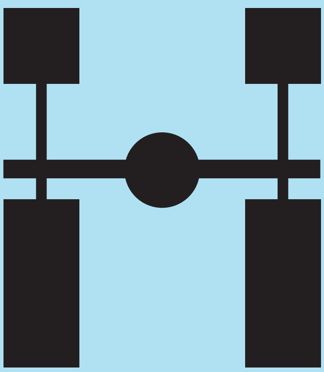
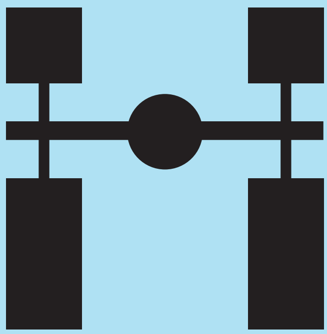


Fig. 8 - Unbalanced multi cavity mould; Incorrect design



Sprue

The sprue should be as short as possible. Hot sprue bushings are often used to reduce cycle time and eliminate sprue sticking for softer grades.

Tapered sprue extractors with 5 degrees taper or more are preferred. The sprue base should be equal to the runner diameter. 'Z' puller or conical designs are preferred.

Recommended are short sprue bushings. A cold slug catcher should be located at the end of the sprue.

Runners

Type
Full round runners or trapezoidal shapes are preferred, half round runners should be avoided.

Size
The size of the runners should be as short as feasible. They should be big enough to minimize the pressure drop in the melt before it enters the mold cavity.

- The primary runner diameter should be equal to the sprue base.
- The secondary runners should have a smaller diameter.

Cold slug catchers should be placed at the extremity of each runner.

Gates

Type
Cylindrical or trapezoidal gate sections are preferred.

It is advisable to adapt the type of gate to the part, e.g.:

- circular parts should have, if possible, a centered gate
- submarine or tunnel gates are recommended for small parts

Various types of gates are shown in figure 9.

Size
The gate size should be as small as possible, typically 1mm in diameter.

- For a part weight less than 10 g, choose a gate diameter of 0.6-1.0 mm or less.
- The gate diameter should be small in order to generate shear at the entrance of the cavity.
- Too large or too small gates could lead to un-aesthetic traces on the part.
- The gate land length ("the tip") should be kept to a minimum; maximum half of the gate diameter.

Number of gates
For small parts one gate is sufficient. For larger parts two or more gates are recommended. Round parts may require more gates to maintain concentricity.

The optimal filling of the part is besides depending on the number and size of the gates, influenced by the material type, gate thickness, melt temperature injection speed and spiral flow length (see figure 1 and 2).

Tips for gate design

- For complex parts, the use of a prototype tool is recommended.
- It is advisable to start with small diameter gates, which can be easily increased if necessary.

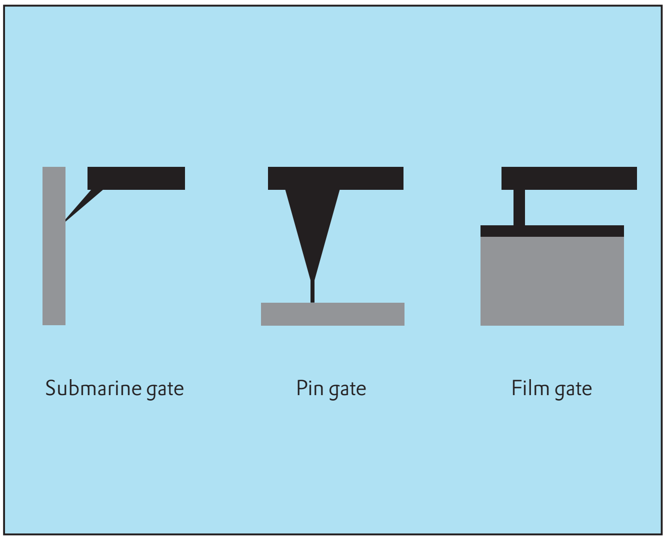


Fig. 9 - Examples of different gate designs

Ejectors

Softer grades of Sarlink require more ejectors than harder ones. Sarlink parts with small undercuts can be easily demolded. A conjunction of ejectors pins and air ejection will facilitate demolding.

Type
Prefer large ejectors to avoid piercing for soft grades. Blade ejectors can be used. Air assisted ejectors are useful for parts with large undercuts.

Location
The ejectors should be spread evenly around in order to balance the ejection and to avoid distortion of the part.

Venting

Sarlink molds must be vented in order to allow for the escape of the air present in the mold when the molten material fills runners and cavities.

Inadequate venting can lead to incomplete parts, burn marks, flash, poor surface appearance or weak welding lines. On the contrary, well vented tools will allow faster cycles.

Vents should be located on the mold parting line, in the area(s) furthest from the gate(s) and at the weld lines. Their depth should not exceed 0.025 millimeter for the part which is in contact with the cavity. This will prevent the formation of flash and the escape of Sarlink material through the vent. At a short distance from the cavity (10-15 mm), the depth of the vent can be increased to 0.15 mm or larger to facilitate the escape of the air (see figure 10).

As previously mentioned, large runners should also be vented. Besides the venting of the cavity, it may also be helpful to add venting on the ejector pins, as depicted in figure 11.

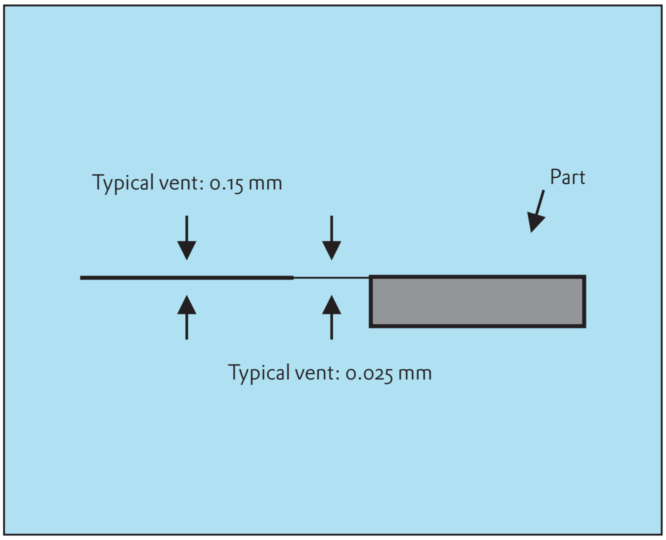


Fig. 10 - Example of venting

Mold Surface

Avoid all polished surfaces. It is preferred to have a medium EDM finish. Vapor honing or bead blasting is also recommended.

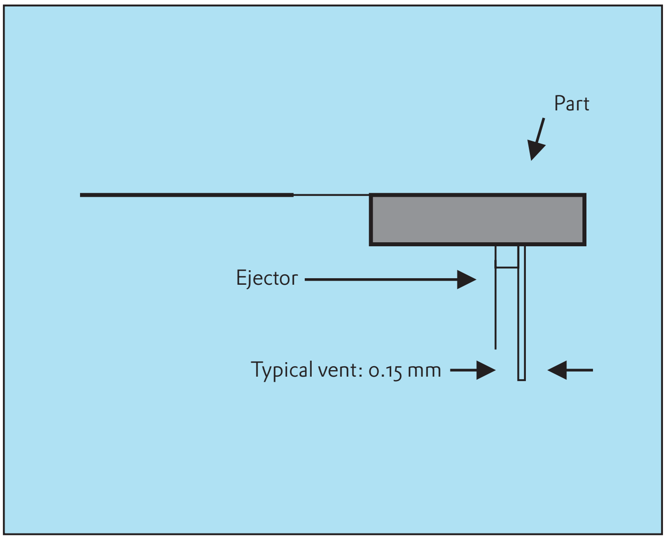


Fig. 11 - Example of ejector pin venting

Mold material

In appendix 2 the recommended types of pre-hardened steels and fully hardened steels are listed.

Aluminum or pre-hardened steel tooling can be used for prototyping and for short production runs.

Shrinkage

Shrinkage is a critical item to control the dimensions of the finished part. It is also a particularly complex parameter, as it can be influenced by many interrelated factors, such as:

- the machinery
- the mold design
- the part geometry and the direction of flow
- the Sarlink grade used
- the processing conditions
- the injection molding system (e.g. hot runners)

Influence of part geometry and direction of flow on shrinkage

The geometry of the part, and its thickness, play an important role to determine the shrinkage. Generally, in the case of Sarlink, the shrinkage is higher in the flow direction than perpendicular to the flow direction.

For shrinkage data on a specific Sarlink grade, contact your Account Manager.

Influence of processing conditions on shrinkage

Some of the variables which influence shrinkage may be very difficult to change (machinery, part geometry, Sarlink grade, etc.). Processing conditions, such as holding pressure, have an influence on shrinkage but can be varied.

- Holding pressure: higher and longer holding pressures will result in less shrinkage.
- Mold temperature: higher mold temperature creates higher shrinkage.
- Melt temperature: the melt temperature of Sarlink does not have a big influence on shrinkage.
- Injection speed

Summary

Reduce shrinkage by:

- increasing the injection speed
- increasing the holding pressure
- reducing the mold temperature
- increasing the mold cooling time.

Increase shrinkage by reversing the above mentioned actions.

Multi-Shot (2K) Injection Molding

Multi-shot injection molding or also called 2K injection molding is the technology where two or more different materials are injected sequentially or together at the same time in a special mold. In this process it is necessary to create a separate cavity space for each material in a single mold. This is accomplished by:

- translation of the entire mold
- rotation of the entire mold
- a replacement of a part of the mold by translation or rotation.

For multi-shot injection molding the injection molding machine must be equipped with a separate barrel for each material. These barrels can be standing parallel or perpendicular depending on the layout of the machine.

Adhesion

Important factors for a good connection are:

- the compatibility of the materials
- the design of the product
- molding conditions.

Due to the fact that Sarlink is based on polypropylene and EPDM rubber, the adhesion between Sarlink and polypropylene is excellent. Normally the hardest material is injected first followed by the softer material.

If the materials are not compatible it is possible to create a connection via mechanical interlocking design.

Molding conditions

Elevated mold and melt temperatures aid in the adhesion of Sarlink to the substrate material.

Recycling

Multi-shot products of Sarlink and polypropylene can be recycled easily. Recycled material which is a mixture of Sarlink and polypropylene is compatible with polypropylene and/or Sarlink. The composition of the multi-shot part will influence the properties of polypropylene or Sarlink.

Hot runner technology

Sarlink can be injection molded using hot runner technology. This technology reduces scrap generated during processing by eliminating cold runners.

In addition to the general information provided previously, the following specific recommendations will help obtaining optimal results when molding Sarlink with hot runners:

- internal heated runner systems are not recommended
- small runner channels are used to keep the shear rate high throughout the runner system
- each runner should be equipped with an individual temperature control, located as close as possible to the gate
- runner should be as short as possible to reduce the residence time
- gates should be smaller than the gates for cold runner molds
- balanced runners are strongly recommended.

sarlink

Appendix 1

Recommended Mold Materials
Steel types moulds

	A prehardened NI-Cr-Mo steel, supplied at 200-330 Brinell, with excellent polishing and photo-etching properties. Suitable for a wide range of injection moulds, blow moulds, extrusion dies	A prehardened stainless holder steel with excellent machinebility, high tensile strength end good corrosion resistance	A through-hardening stainless steel with good corrosion resistance and very good polishability	A versatile through-hardening 5% Cr-mould and die steel with good wear resistance and polishability
Properties [1]	**I 45NI-Cr6**	**II X36CrMo17**	**III X40Cr13**	**IV X40CrMo51**
Normal hardness [2]	**(-310)**	**(-340)**	**52**	**52**
Wear resistance	■■■□□□□□□□	■■■■□□□□□□	■■■■■■■□□□	■■■■■■■□□□
Toughness	■■■■■■■■■□	■■■■■■■□□□	■■■■■□□□□□	■■■■■■□□□□
Compressive strength	■■■■□□□□□□	■■■■■□□□□□	■■■■■■■□□□	■■■■■■■□□□
Corrosion resistance	■■□□□□□□□□	■■■■■■■■□□	■■■■■■■■■□	■■■□□□□□□□
Machining [3]	■■■■■□□□□□	■■■■■■□□□□	■■■■■■■■□□	■■■■■■■■■□
Polishability	■■■■■■■■□□	■■■■□□□□□□	■■■■■■■■■□	■■■■■■■■□□
Weldability	■■■■□□□□□□	■■■■□□□□□□	■■■■□□□□□□	■■■■□□□□□□
Nitriding	■■■■■■□□□□	□□□□□□□□□□	□□□□□□□□□□	■■■■■■■■■■
Photo-etchability	■■■■■■■■■□	■■■■□□□□□□	■■■■■■■■□□ [4]	■■■■■■■■□□ [4]
Norms				
DIN	**(1.2710)**	**(1.2316)**	**(1.2083)**	**(1.2344)**
USA AISI	**P20**	**420F**	**420**	**(H13)**
Delivery HB	**~310**	**340**	**215**	**175**
Analysis (3/4)				
C	**0.37**	**0.36**	**0.38**	**0.40**
Cr	**2.0**	**17.0**	**13.6**	**5.2**

1) The properties of steel grades have been rated from 1-10, where 10 is the highest rating. Such comparisons must be considered as approximate, but can be a useful guide te steel selection.
2) Special process required.
3) Rockwell C (Brinell).
4) Tested in delivery condition.

Source properties: Steels for Moulds, Uddeholm

Appendix 2

Summary of Recommended Settings for Optimal Injection Molding of Standard Sarlink Parts

Temperature settings (°C):	
Rear barrel	165 - 175
Middle barrel	180 - 200
Front barrel	200 - 220
Nozzle	210 - 230
Melt	200 - 230
Mould	10 - 60
(Hydraulic) pressure settings:	
· Injection pressure	60 - 120 bar
· Holding pressure	50 - 70% of injection pressure
· Back pressure	5 - 20 bar
Speed settings:	
· Injection speed	85% of maximum
· Screw speed	50 - 150 rpm
Time settings:	
· Injection time	0.5 - 2 s
· Holding time	1 - 10 s
· Cooling time	As short as possible. The part should be removable without deformation or piercing of the ejector(s)

Typical melt temperatures for Sarlink in a multi-shot process

Material 1	Material 2	Tmelt 1 (°C)	Tmelt 2 (°C)	Tmould (°C)
polypropylene	Sarlink 3000/4000	210 - 240	210 - 250	40 - 80
polyamide	Sarlink 3500	230 - 280	260 - 290	70 - 90

sarlink

Appendix 3

Troubleshooting Table for the Injection Molding of Sarlink

Problem	Cause	Solution/corrective action
Short shot	Insufficient filling of the mould	Increase shot size (normally fill to 98% of total volume)
		Increase injection speed/pressure
		Decrease hopper-side temperature to improve plasticizing
		Check operation of spare ring
		Increase back pressure
		Check venting
		Check mould temperature
		Check hot-runner temperature/controllers
		Check balance of flow for hot-runner gate systems
		Enlarge small passages in the mould
	Mould functioning problems	Check working of movable parts in the mould, especially for multi-shot injection moulding
Sink marks	Insufficient filling of the mould; mould is underpacked	Increase shot size (normally fill to 98% of total volume)
		Increase injection speed
		Increase holding pressure and/or holding time
		Increase barrel and nozzle temperature
		Increase back pressure
		Increase gate size and/or heaters
		Increase/check the venting
Weld lines: Strength and visibility	Too low material temperature	Increase melt temperature
		Increase mould temperature
		Increase back pressure
		Increase injection speed/pressure
		Increase holding pressure
	Insufficient venting	Check venting; clean and/or increase size
	Mould design problem	Increase vent size
		Increase runner size
Silver streaks on part	Moisture in material is too high	Dry material
		Check the venting of the mould
	Too cold mould (moisture condensing)	Increase the mould temperature
		Wait for constant production
Colour degradation	Material or colour concentrate degradation	Decrease temperature profile
		Reduce injection speed/pressure
		Reduce back pressure
		Change to higher temperature stable colour concentrate

sarlink

Problem	Cause	Solution/corrective action
Voids and bubbles	Moisture in material is too high	Dry material
		Increase back pressure
		Increase injection speed/pressure
		Increase mould temperature
Part sticks in the mould cavity or in sprue	Insufficient cooling of the part	Reduce injection pressure
		Decrease holding pressure
	Incorrect mould filling	Add decompression
		Increase mould filling volume
		Increase injection speed
		Increase melt temperature
		Increase back pressure
	Incorrect functioning of the mould	Check and/or clean the venting of the mould
		Clean mould and cavity
		Check heating of hot runner cavities
Poor surface appearance	Incorrect mould design	Increase gate size (reduce shear)
		Decrease the surface smoothness (add a grain)
	Problem with additives	Stop with using mould lubricants
		Basis of (colour/UV) masterbatch is not correct
	Too high moisture content	Dry material
Distorted parts	Improper removal operation	Decrease ejection time
		Change to larger ejection pins
		Eject on strongest area of the part
		Increase cooling time
		Reduce melt and/or mould temperature
	Stresses are moulded into the part	Increase melt temperature
		Reduce filling time/pressure
		Reduce holding pressure
Adhesion problems in a multi-shot process	Temperatures are too low	Increase ('contact temperature') melt temperature of materials
		Start with increasing the temperature of the first injected material
		Reduce switch time between two materials
		Increase holding time/pressure of second material
	Materials are not compatible	Check if the material is compatible with the Sarlink grade in use
	Negative influence of adding of additive	Too much adding of an incompatible ingredient to the Sarlink grade used can lead to adhesion problems