Find out more about
your TPE solutions
partner now.

elastron.com
elastron@elastron.com

elastronusa.com
elastron@elastronusa.com

ENGI-NEERING LIFE

PROCESSING GUIDE FOR INJECTION MOLDING

elastron

ENGI-NEERING LIFE

Science has made us understand the world. However, engineering has changed the world by taking knowledge and putting it into practice. It was engineers who wrote history and integrated knowledge into life.

Elastron, a global industrial company, is a daily life engineer with solutions that touch and improve many aspects of life. Although it produces high-tech thermoplastic elastomers for industrial companies, its expertise continuously improves life through different products and services.

Elastron is not just a competent manufacturer. Elastron produces to enhance everyday life.

Everyday Life Engineering Partner of Global TPE Users

Elastron is the developer of products that touch human life and make everyday life easier by using thermoplastic elastomers. Elastron is the world's TPE specialist. It always aim to improve life.

GLOBAL

Elastron has a global service network that can serve more than 55 countries.

SOLUTION PARTNERSHIP

Elastron has a business-service partnership philosophy beyond product supply.

PROACTIVE

Elastron goes beyond the customer's current needs by combining its worldwide product, service and knowledge with its agile structure.

About TPE

TPEs are rubberlike materials that can be processed on any plastic machinery. They enable higher savings on processing costs compared to vulcanized rubbers, consequently increasing company's profits. Although they have functional characteristics similar to vulcanized rubbers, the production and investment costs are lower.

TPEs offer low density, wide hardness range, weathering and temperature resistance, recyclability, good compression set and easy coloring.

- Temperature Resistance
- Weathering Resistance
- Recyclability
- Wide Hardness Range
- Easy Coloring

Developer of Products That Touch Human Life

Elastron's main production facility in Gebze, Turkey, is at the intersection of the East and the West. It offers logistical advantages with land, sea, and air routes.

Elastron second production facility is in Gainesville, Georgia, USA and serves the North American market with warehouses in the East and the West Coasts of the United States.

Elastron has offices in China, Taiwan, and Japan for the Asia Pacific market. It also uses a Germany office to provide solutions to the European market.

Elastron's Quality Approach

Elastron devotes significant resources to technical developments of new products and applications to meet market demands and customer needs. A core value of Elastron is superior customer service, and significant resources are devoted to ensure maximum customer satisfaction.

Elastron has obtained all the key quality management systems. The Quality Management System (ISO 9001:2015) ensures that Elastron supplies all the products with consistent quality that meet customer requirements.

Elastron is also accredited by the Automotive Quality System (IATF 16949:2016) ensuring high quality production and supply to the automotive market.

Occupational health and safety practices of Elastron are certified with ISO 45001:2018.

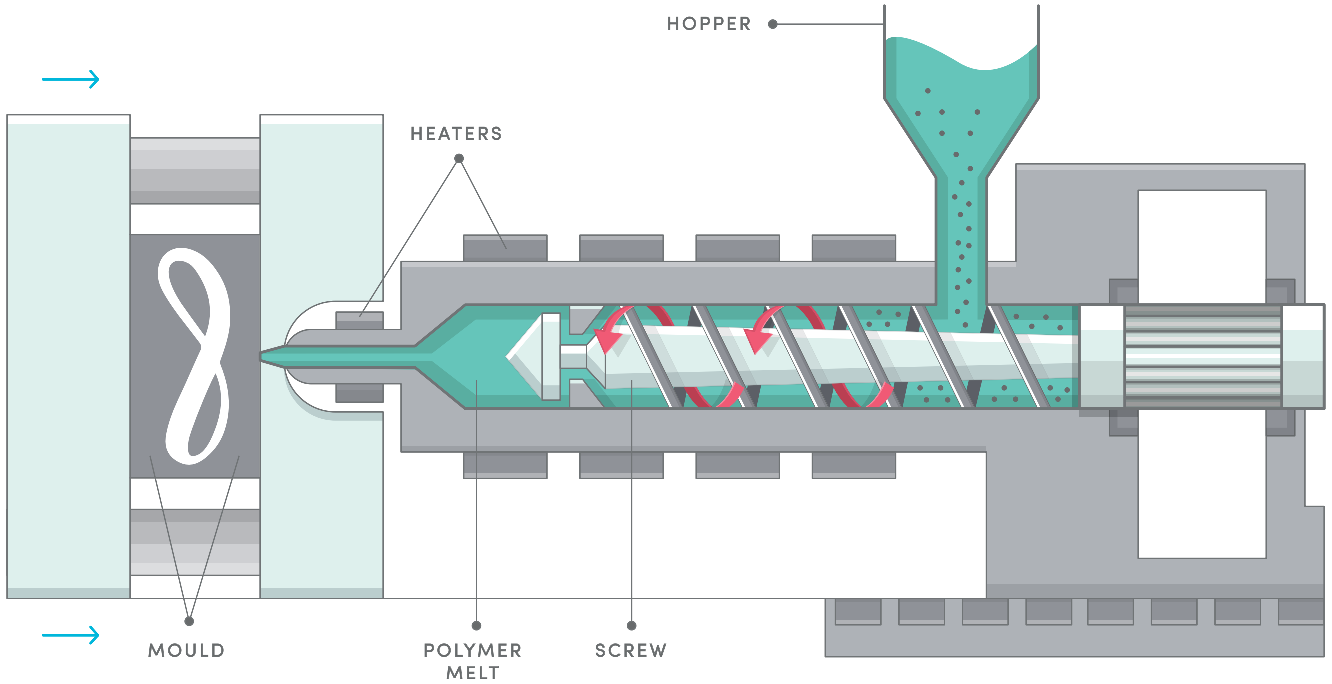
The latest achievement is the certification of Elastron according to ISO 14001:2015 Environmental Management System, which demonstrates our strong commitment to the environment.

Recently Elastron has expanded its global reach with special focus on North America and China. Today Elastron is serving over 55 countries with high quality products that meet customer needs.

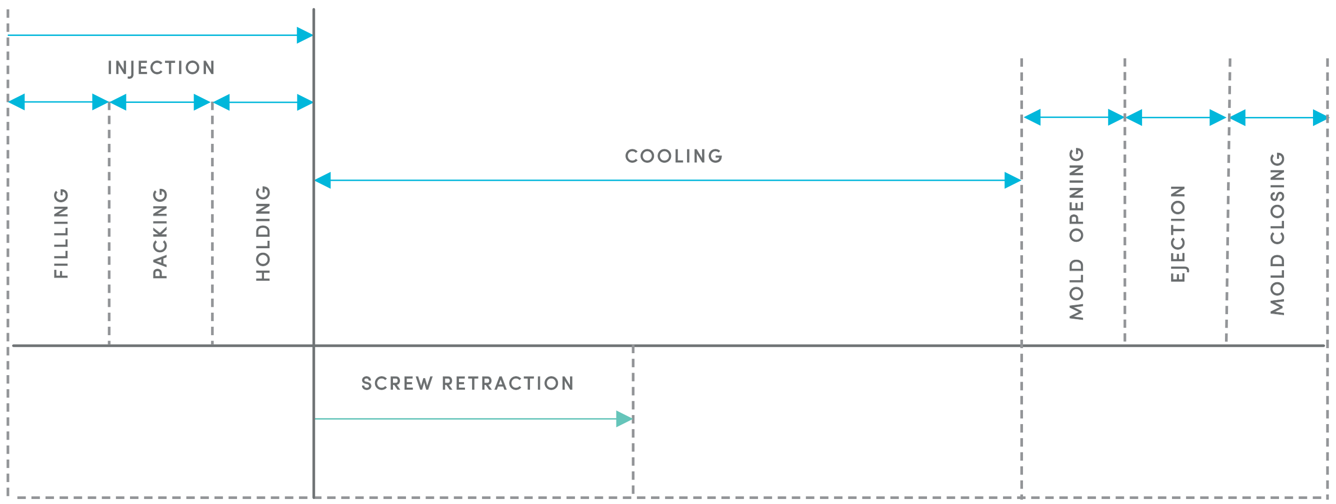
Elastron R&D developes customer-oriented solutions in addition to its high-quality general product line.

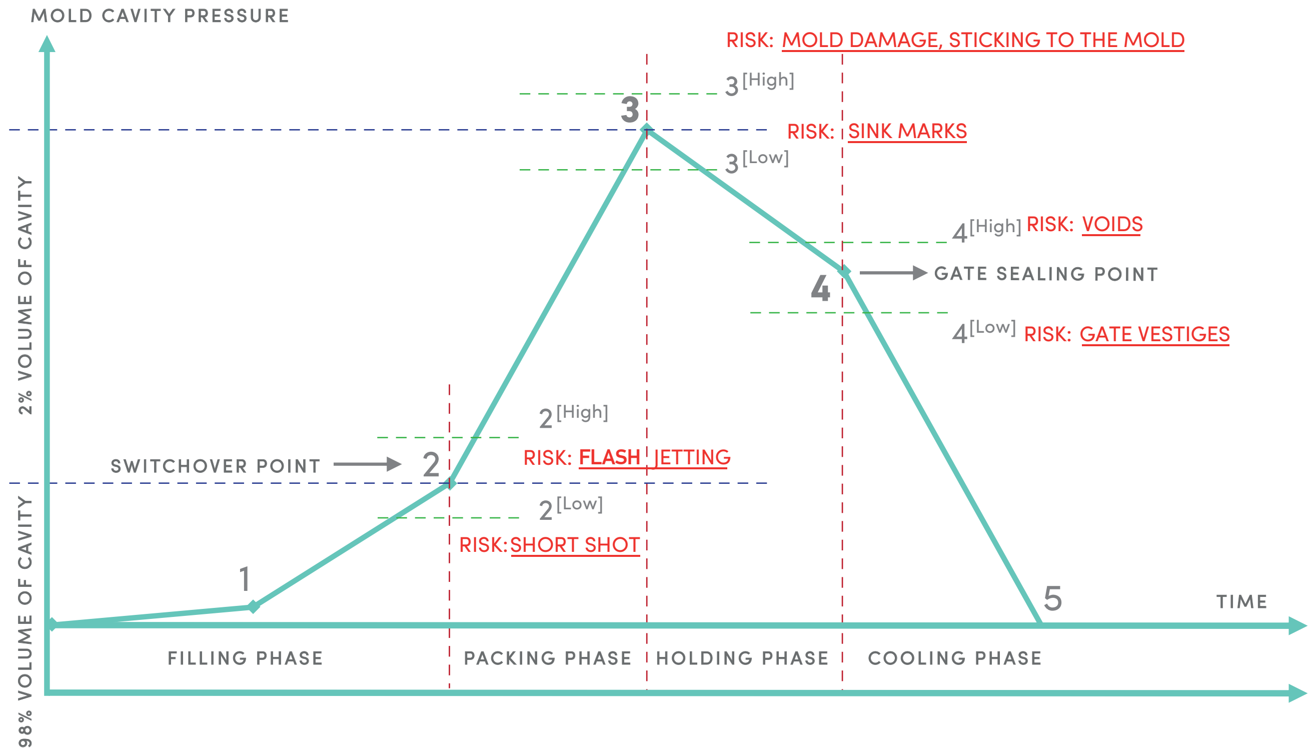
INJECTION MOLDING

An injection molding press consists of two halves, a clamping side and an injection side. The main purpose of the clamping side is to open and close the mold and the ejection of the molded parts. The purpose of the injection unit is to melt the material by controlled heating and then to inject molten material into a mold. The screw rotates with the rpm specified on the screen to melt material introduced from the hopper and to accumulate molten material in front of the screw. After the required amount of molten material is accumulated by the screw retraction, the injection phase begins. As molten material is introducing into the mold, the injection machine controls the ram speed of the screw or injection speed (speed profile). In addition to this, it controls the molding pressure after molten material fills into the cavities.



INJECTION MOLDING CYCLE





1. Filling Phase
The melt must be filled to the mold as fast as possible according to the part geometry. If there are any restricted areas affecting the filling properties, the proper speed profile must be determined. The flow characteristics are determined by the melt temperature, the speed, and the shear rate. An injection speed that is too high can create excessive shear and result in defects, such as splay and jetting. High speed filling terminates at the Switchover point, and the low speed packing phase commences.

1.1. Switchover Point
It is a transition point from filling to packing, and plays a crucial role in the quality of the molded parts. Late switchover can cause building up of excessive cavity pressure, and it leads to flash and mold opening. Early switchover leads to short shots and longer cycle times.

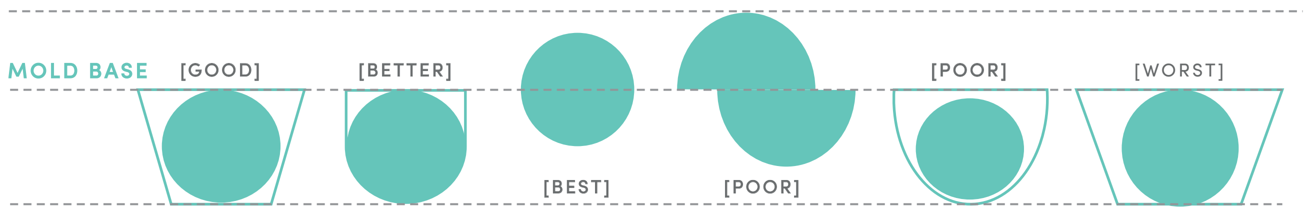
2. Packing Phase
As soon as the material gets into the cavity, cooling starts and it induces shrinkage. That is why there is a need to inject more material in order to prevent this shrinkage. After loading to 98% of the volume of cavity, the injection speed and pressure reduces during this phase. The packing pressure determines the part weight and part dimensions. It is important to completely fill the mold and avoid over packing 3[High] or under packing 3[Low].

3. Holding Phase
By reducing the pressure and applying speed, a certain amount of pressure must be applied until the gate completely freezes. Gate freezing is the solidification of material around the gate area and prevents the melt from back flowing out of the mold. This keeps the dimensions and weight values stable

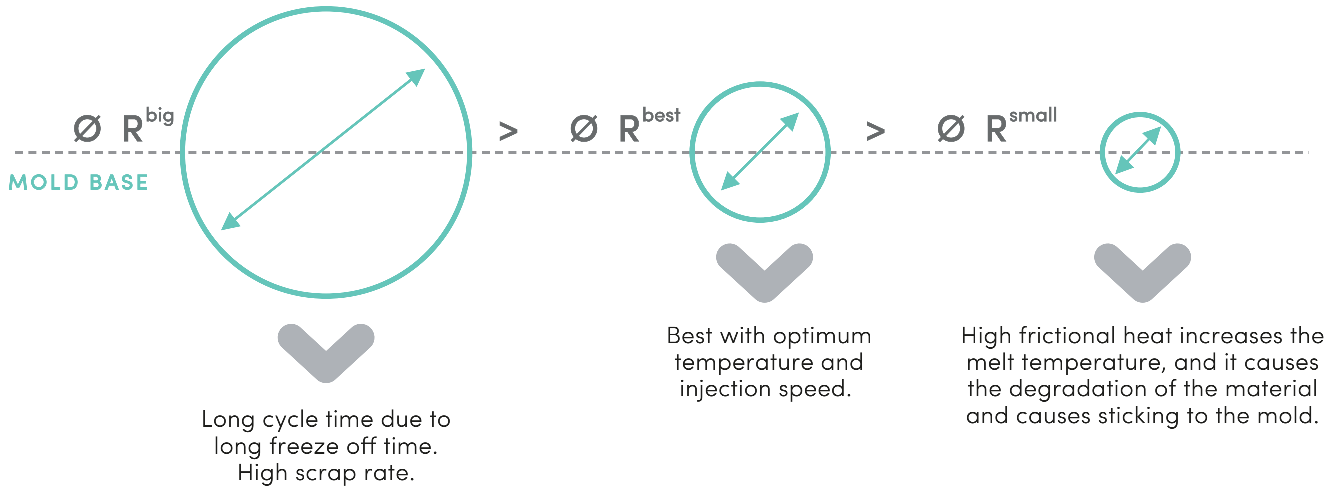
4. Gate Sealing Point
The plastic enters the cavity through the gate. As long as the gate is not frozen, new material comes in to the mold replacing unoccupied free spaces left from shrinking material. At the end of the solidification, the gate is completely blocked due to cooling and no material can get back to the flow path.

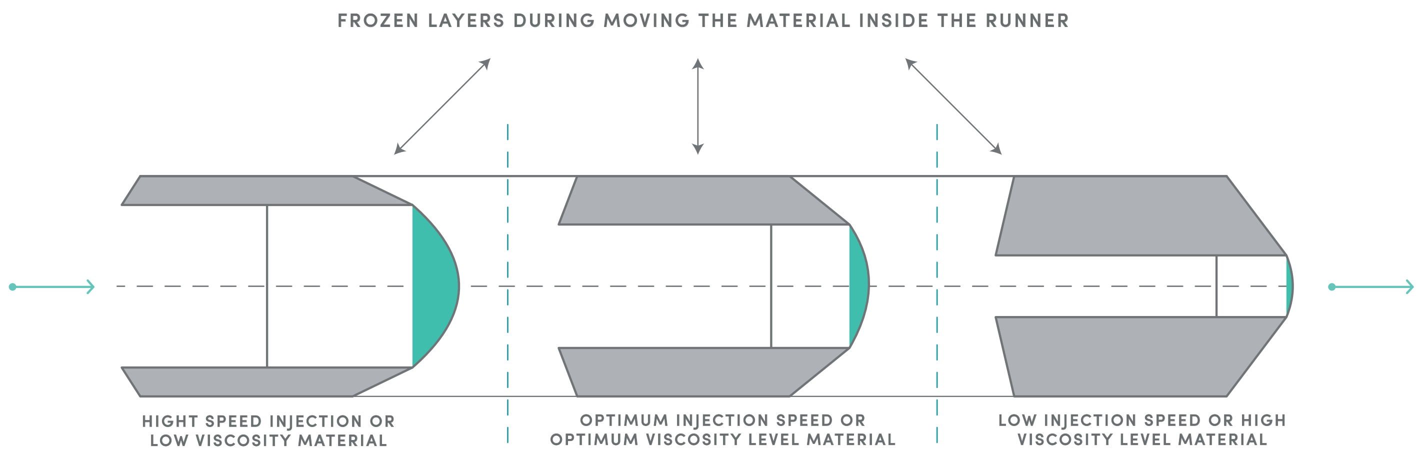
5. Cooling Phase
To ensure optimum molding cycles while maintaining part surface requirements and mold filling capability, a stable mold temperature should be determined and maintained. The cooling time takes about 80% ofthe injection molding cycle. A well-designed cooling system can shorten the molding time and improve productivity.

MATERIAL MOVING IN THE RUNNER



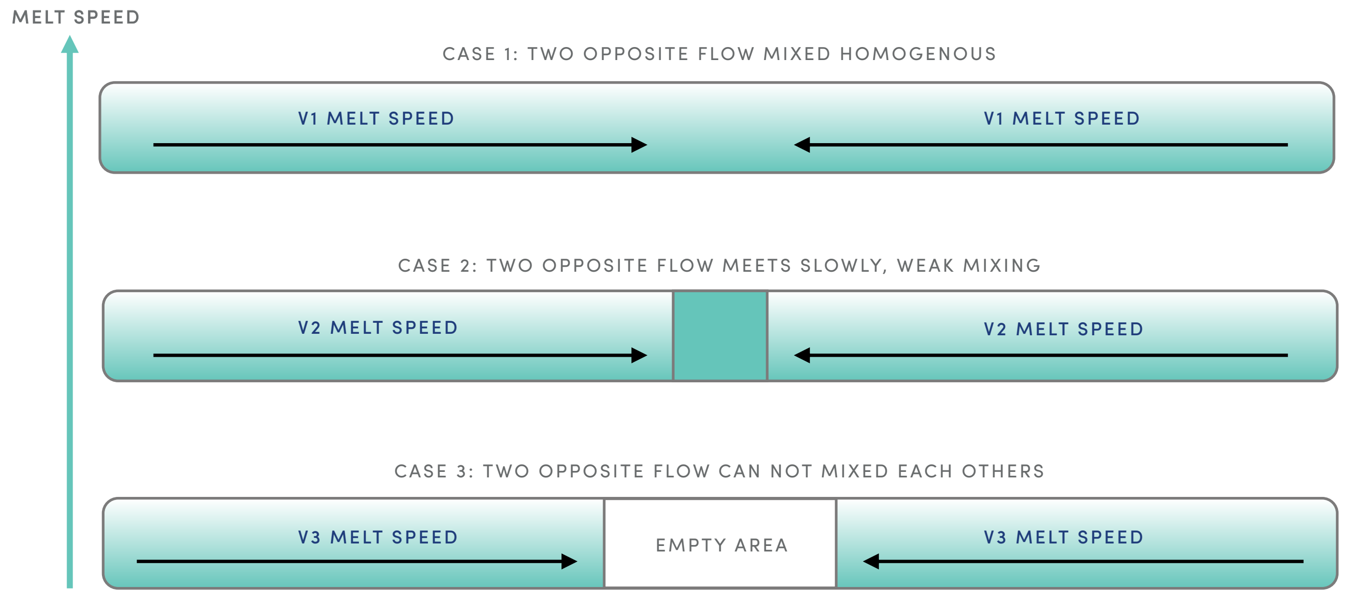
As shown above, the best selection on the runner is fully rounded and centered to the mold base.





During injection, the injection speed and material viscosity are two of the most important factors to effect the molding quality. If the material is a low viscosity material, do not use an excessive injection speed. High speeds generate high shear rates, and the material gets hot on the runner and the temperature will exceed the degradation level. In this case the material would adhere to the cavity wall after injection. If the material is sticking to the mold, reduce the injection speed gradually.

MATERIAL MOVING IN THE MOLD



Case 1
Due to the optimum melt speed, the two opposite flow channels meet seamlessly without any weak areas.

Case2
The same material with a low melt speed causes weak mixing. **Cold Weld Line.**

Case 3
The same material with very low melt speed causes an empty area due to not joining flow fronts. **Short Shot.**

WHY THE MELT SPEED IS LOW

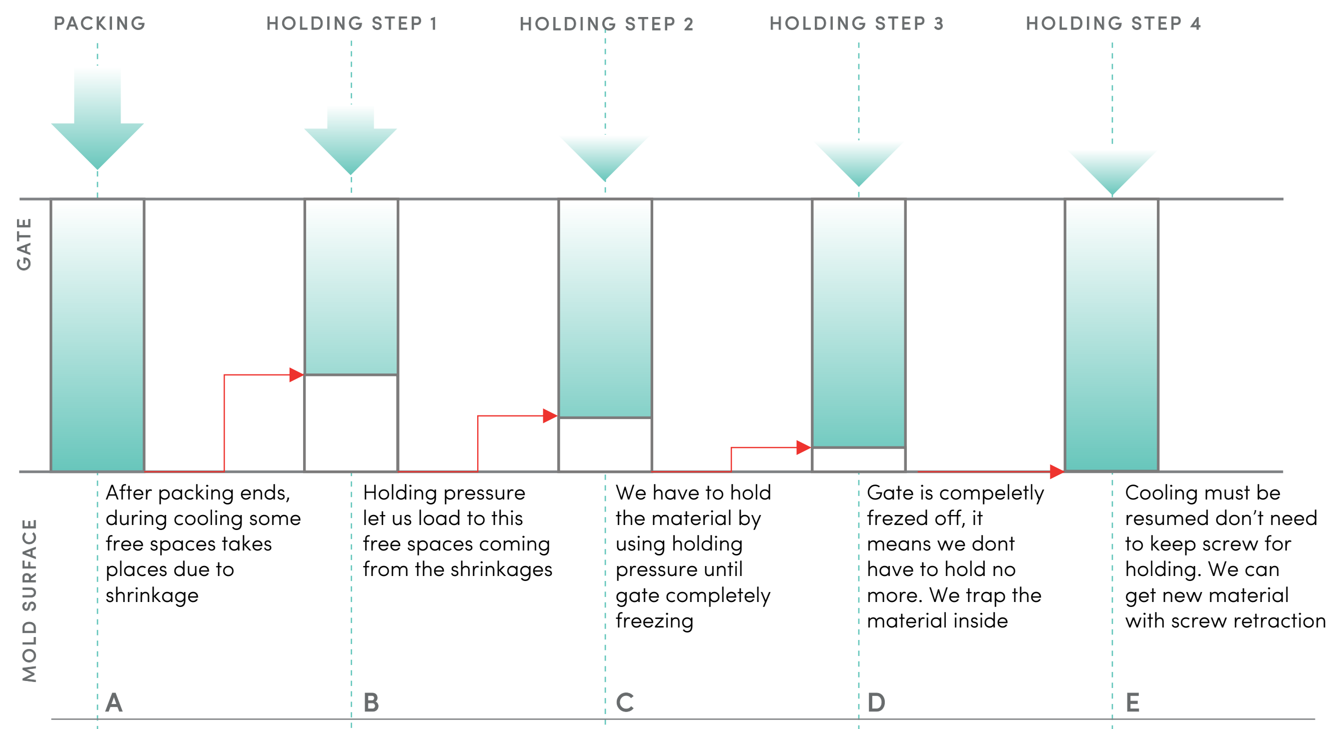
• **Low injection speed**
During the filling phase while the injection goes to the cavity through the flow path, the melt cools over time. This is why the material must cool as fast as possible. Otherwise the melt cools prematurely, and there will not be homogenous mixing at the intersection line of the two separate flows.

• **High viscous material**
The material viscosity might be higher than normal. In this case the material runs slowly in the flow path despite the injection speed increasing.

• **Low mold temperature**
If the mold temperatures are lower than normal, the material cannot flow in the mold. It can make a wavy surface on the part.

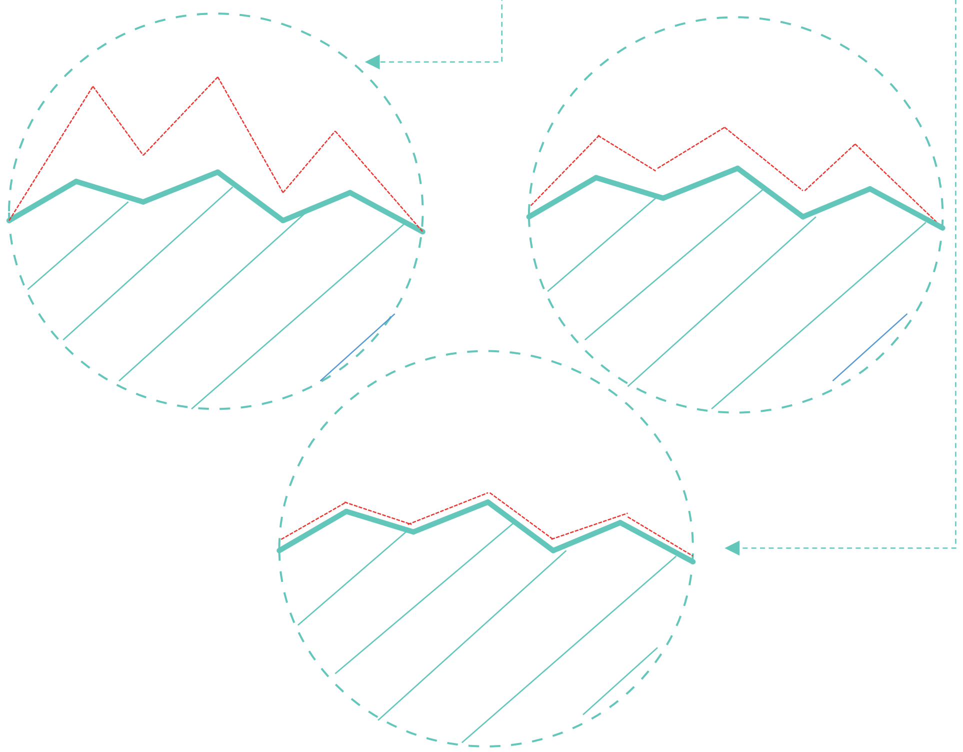
• **Improper gate location**
Gate
• **Low gate diameter**
Causes injection speed pressure loss in the cavity

• **Insufficient melt temperature**
A lower melt temperature causes higher viscosity



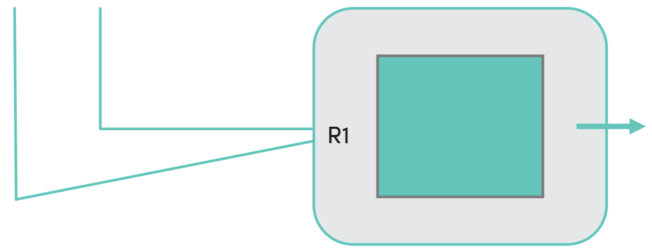
If we don't apply holding or insufficient holding leads

- Sink Marks
- Short Shot
- Surface gloss difference

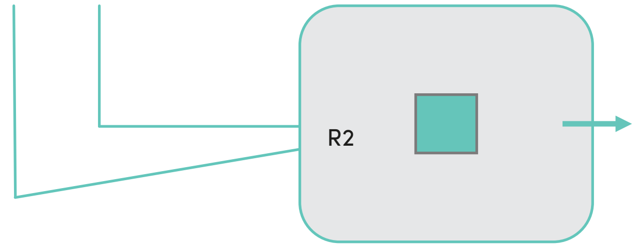


WHY GATE DIAMETER IS IMPORTANT DURING GATE SEALING

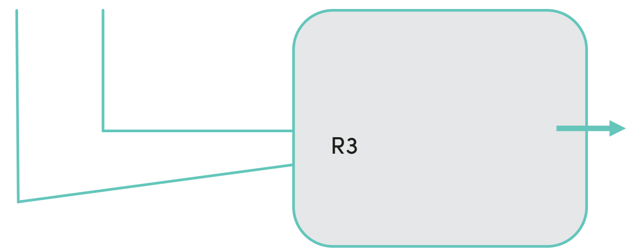
Hold the material after the packing phase until the gate is completely frozen. Otherwise the liquid material goes back inside the runner over the gate. This is why the gate diameter is so important because it must freeze completely. A larger gate diameter increases the holding time. A lower gate diameter leads to premature freezing.



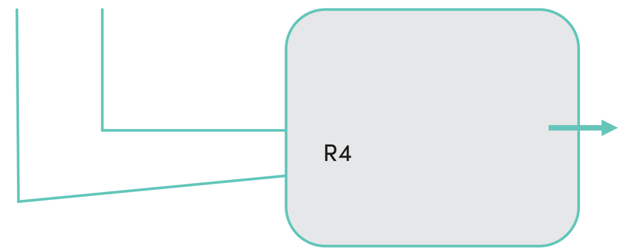
The gate diameter is lower than normal. That means the material is still hot enough even if the gate is completely sealed. Due to the lower diameter, the gate froze earlier than normal, and the press must push some material when the hot material cools down. In this case after injection, there would be voids inside and at the same time some areas would collapse.



The gate diameter is still not sufficient because there are some areas that must be cooled. Increase the diameter gradually.

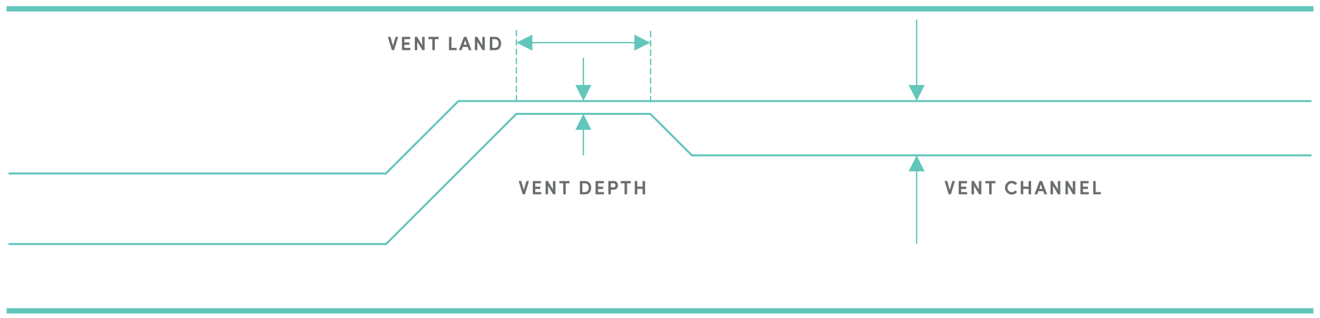


This is the best diameter. There would not be any warpage because, after gate sealing, there are no hot areas.



A higher gate diameter increases the gate sealing time so the cycle time will be greater.

VENT DESIGN

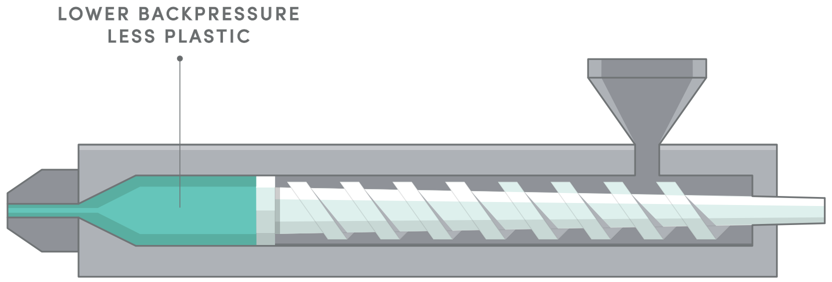


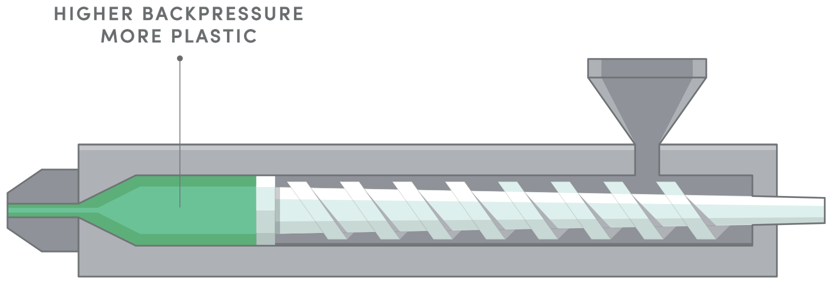
Vent Depth > Normal → Flash, Material sticks to the mold

Vent Channel < Normal → Gas cannot be removed completely.

WHY BACK PRESSURE IS IMPORTANT

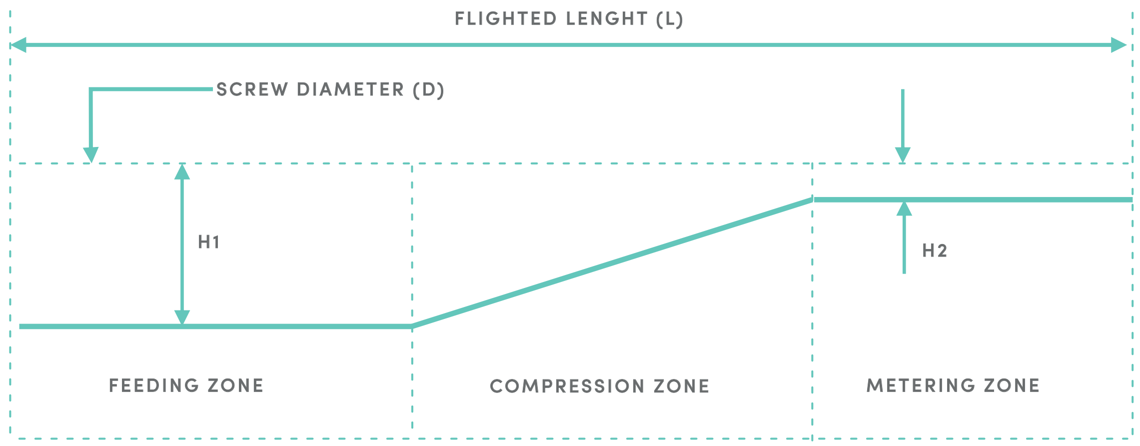
Back pressure is the pressure on the front of the screw during screw retraction to prepare the next shot. Increasing the back pressure collects more material in front of the screw. If the injection speed strokes are not changed, the part weight will be heavier than before because of the higher compression taking place in the same volume in front of the screw.





elastron

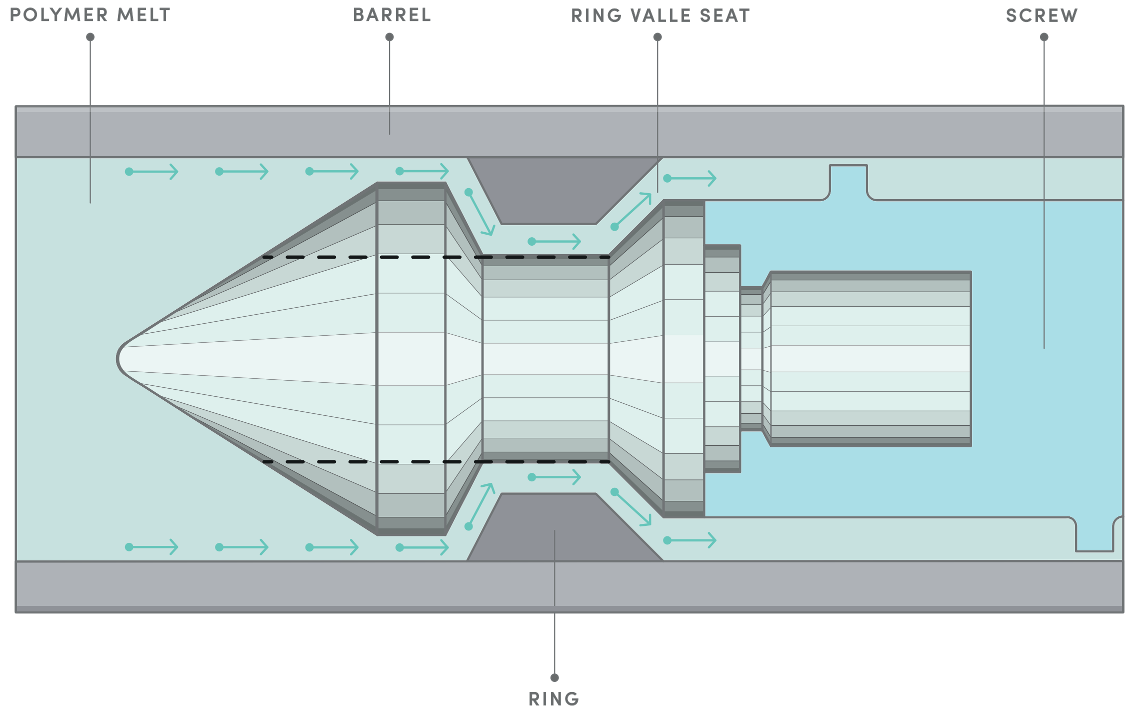
SCREW



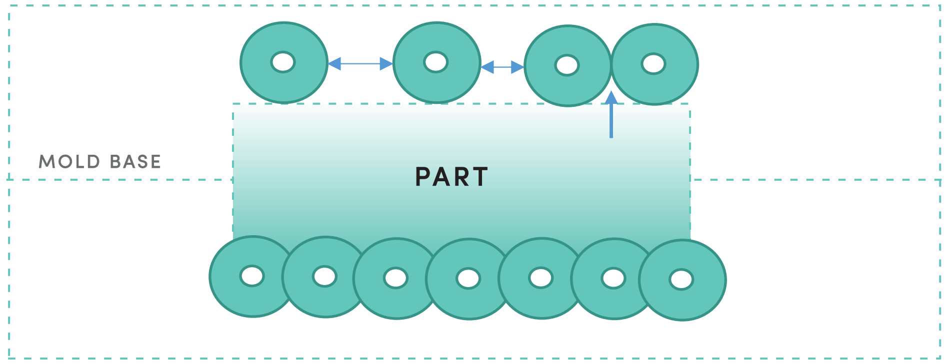
Compression ratio = H1/H2
L/D = Flighted Length / Screw Diameter

Reciprocating screw injection molding machines are equipped with a general purpose screw design and are adequate for processing the material.

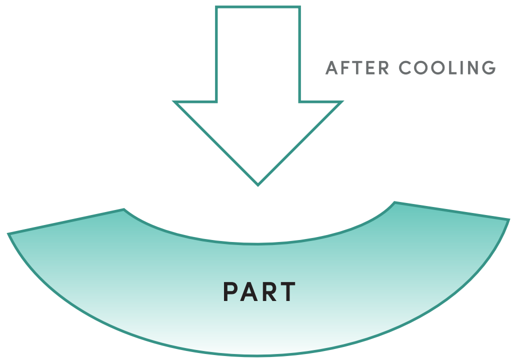
This screw design should have 50% of its length at the feed zone, 25% at the compression zone, and 25% at the metering zone. A screw L/D (length to diameter) ratio between 18:1 and 24:1 and a screw compression ratio between 2:1 and 3:1 are recommended. A floating check ring, rather than a ball check, is also recommended. Nozzles should have a free-flow design, and be as short as possible.



COOLING



The cooling channels must be installed correctly. If there are any voids between them, the cooling effect on the part is not balanced. This is why warpage occurs.



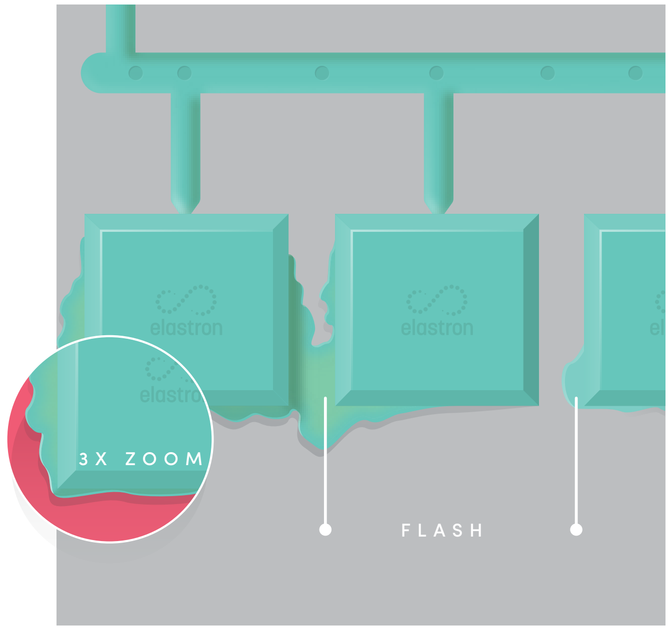
TROUBLESHOOTING

1. Short Shot

- Restricted flow areas, such as the wrong type of gates, the wrong dimension of runners, and thin walls.
- Low melt or mold temperatures.
- Due to insufficient ventilation, air is trapped inside the cavity.
- Insufficient machine injection pressure and low injection speed.
- Machine malfunctioning, such as a blocked feed throat, or a worn non-return valve that causes a loss of injection pressure. There could also be pressure loss on the flow path.
- Premature solidification of the material melt, poor injection speed profile, or prolonged injection time.

2. Flash

- Low clamp force: The clamping force must be higher than the injection pressure. If the clamping force is too weak to hold the mold plates together during the injection process, flash takes place.
- Gap within the mold: Flash will occur if the parting surface does not contact completely, due to a deformed mold structure, parting surface defect, improper machine and mold set up, or a foreign material is stuck on the parting surface.
- Injection parameters: High melt temperature (lowered the viscosity of the material) or high injection pressure with the clamping force and flash takes place.
- Nonconforming venting: Due to the venting design and a very poor venting system, or the venting dimension is not correct.



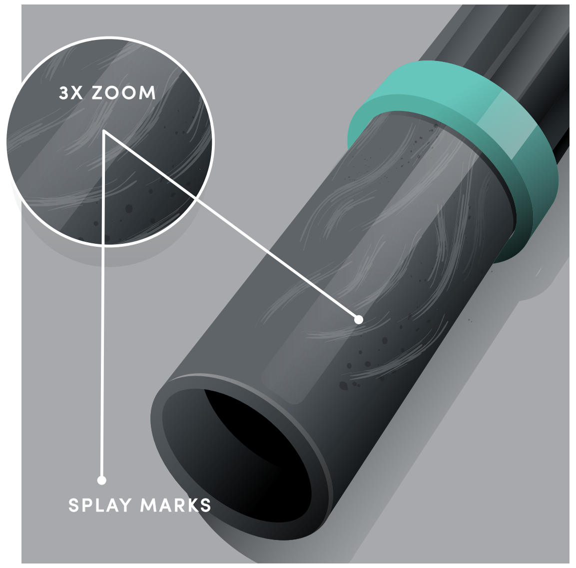
3. Sink Marks

- After injection holding must be applied until the gate is completely frozen. Otherwise the melt goes back inside the runner and sink marks appear.
- Short cooling time, after stress relaxation in cooling, the new material loaded to the cavity replaces the voids.
- High melt temperature or mold temperature
- Due to the mold design, there might be some pressure loss in order to hold the material after injection.

4. Splay Marks

If the material is used without predrying, the moisture inside the material can cause splay marks. A high rpm during screw retraction generates gas and this gas is injected with the parts and melted material.

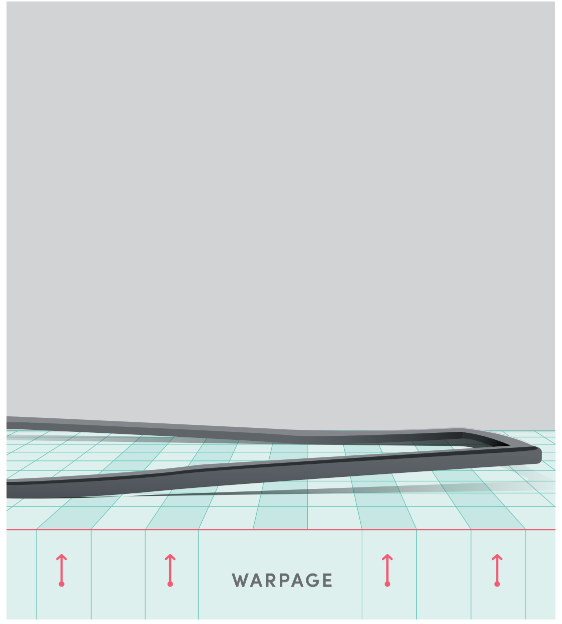
- Dry plastic according to the recommendations.
- Reduce the injection speed, because a high speed generates gas due to friction.
- Reduce the melt temperatures by reducing the barrel temperatures.
- Reduce the screw rpm as a high rpm generates gas.
- Increase the back pressure. Increasing the back pressure will discharge the gas through the hopper.
- Increase the mold temperatures
- Increase venting
- Increase the gate diameters. A tight diameter generates gas due to the high shear rates.



5. Warpage

Warpage is caused by a differential cooling rate of the melt in two sections of the molded product.

- Reduce the melt temperature to reduce the cooling time.
- Reduce the mold temperature to a reasonable level according to the recommendations.
- Increase the pack and hold pressures if there is any pressure loss.
- Increase the pack and hold times.
- Due to an insufficient cooling time, warpage takes place after ejection. Increase the cooling time to cool the part to a sufficient level.

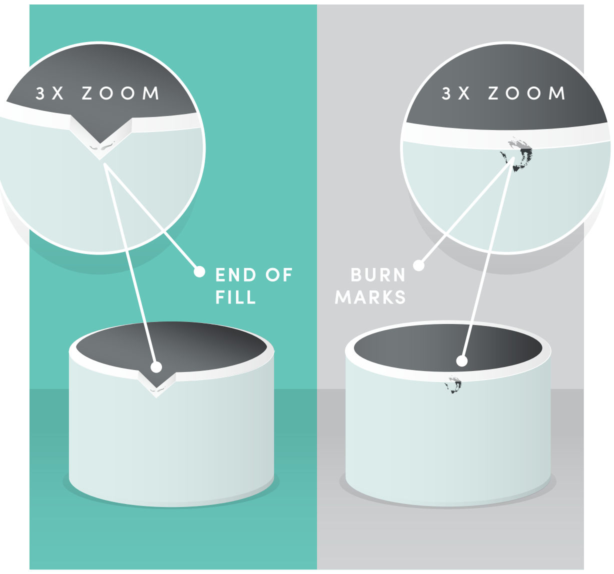


6. Burn Marks

Material degradation

Burn marks can result from degraded materials and appear on the surface of the molded part or near the venting areas. Material degradation is caused by:

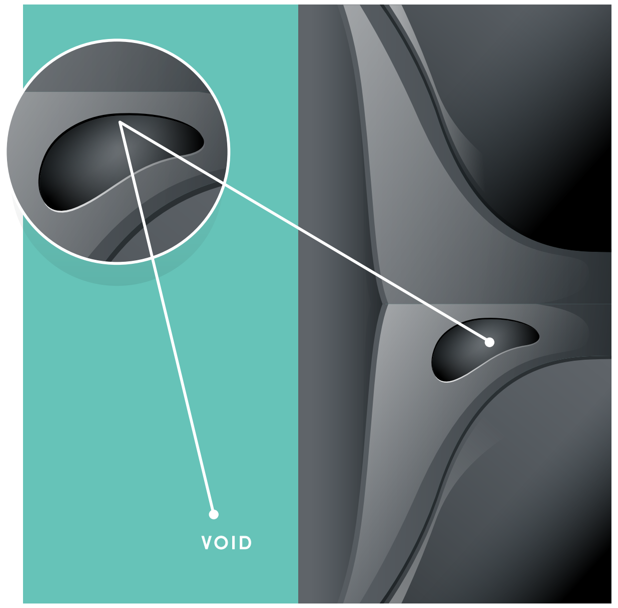
- High melt temperature: An excessive melt temperature is caused by higher barrel temperatures, or a malfunctioning temperature controller.
- High screw rotation speed: If the screw rpm is too high during the plasticization time during screw retraction, it will create too much frictional heat and degrade the material.
- Restrictive flow path: When the melt flows through the flow path with a restrictive nozzle, runner, gate, or part sections, the shear rate increases the heat degrading the material.
- High speed injection: Creates extra heat due to the friction in the runner.



7. Voids

Occur when the parts thickness is high. If the mold temperature is higher than normal during cooling, the plastic melt shrinks towards the wall and therefore pulls a vacuum void on the inside of the part

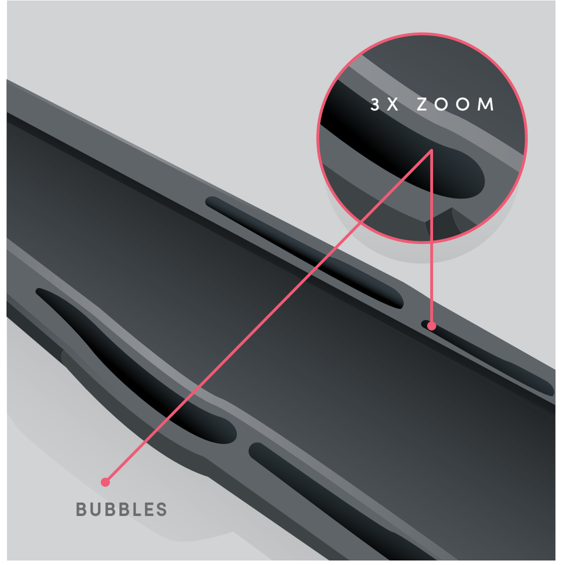
- Reduce melt temperature: Cooling after high temperatures might cause the voids.
- Reduce mold temperature: High mold temperatures keep the melt temperature high causing voids.
- Reduce injection speed: High speed injection creates high frictional shear and melt cooling would be hard.
- Increase pack and hold pressures: Packing and holding pressures send the gas or airs to out through the ventilation channels.
- Increase pack and hold times.



8. Bubbles

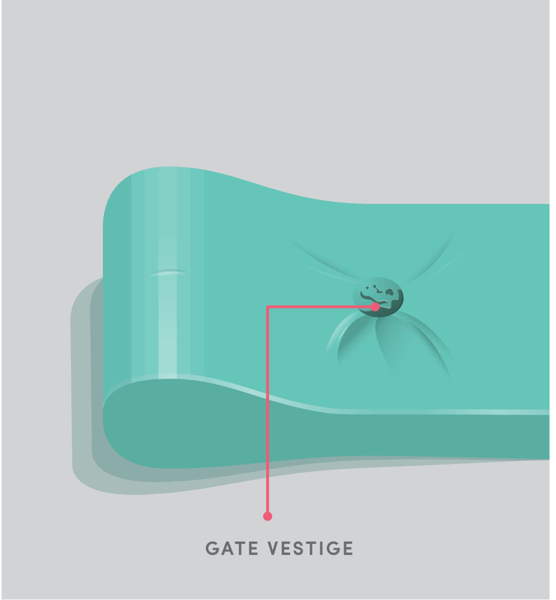
Moisture or gas coming from the product is injected in the mold cavity. This moisture or gas, if embedded inside the melt, can appear as bubbles.

- Dry material to suggested moisture levels.
- Increase back pressure: Back pressure sends the gas to the outside of the mold.
- Reduce the melt temperature.



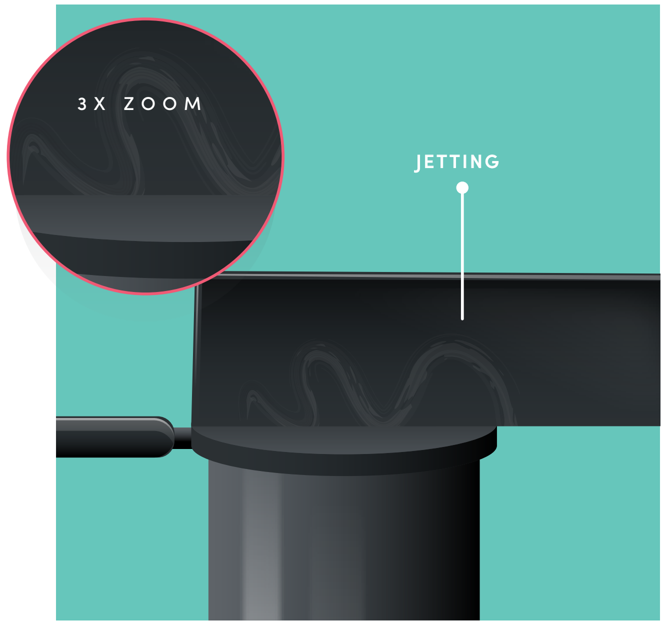
9. Gate Vestige

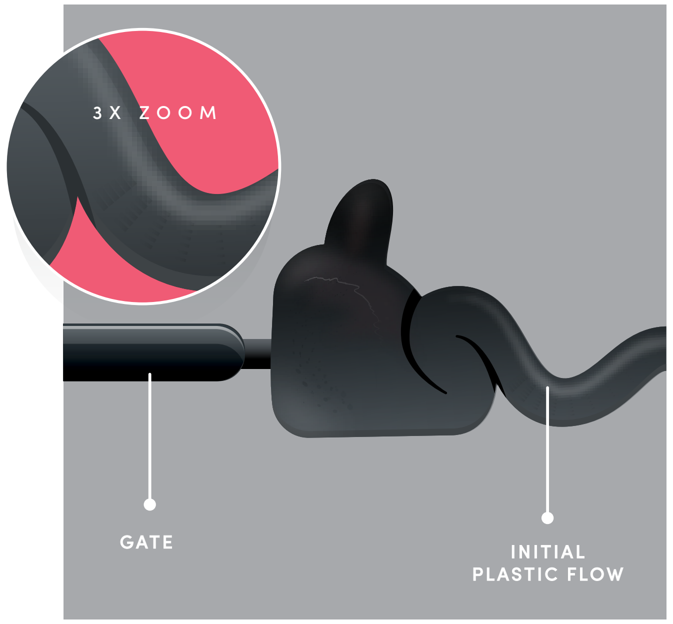
- During the holding phase, reduce the injection speed and pressure.
- Optimize the injection speed profile.
- Increase the melt temperature by increasing the barrel temperatures gradually.



10. Jetting

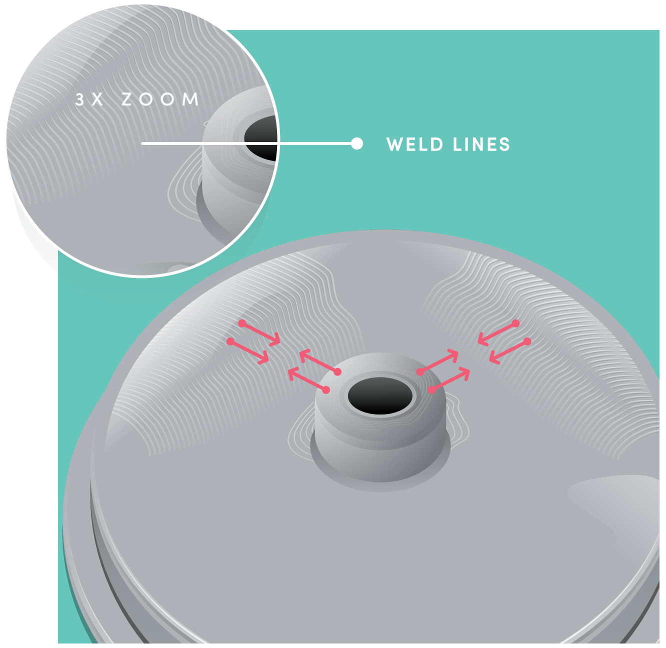
- Place the gate against the metal surface
- Use an overlap gate or a submarine gate
- Slow down the melt with a gradually divergent flow area
- Reduce the injection speed
- Increase the melt temperature





11. Weld Lines

- Increase the melt temperature: If two different flows meet together and are colder than normal, weld lines will appear.
- Increase the mold temperatures: Cold mold walls prevent material flow.
- Increase the injection speeds: Lower injection speeds cause premature cooling.
- Increase the venting: Ventilation supplies better flow in the mold.



INJECTION MOLDING RECOMMENDATIONS

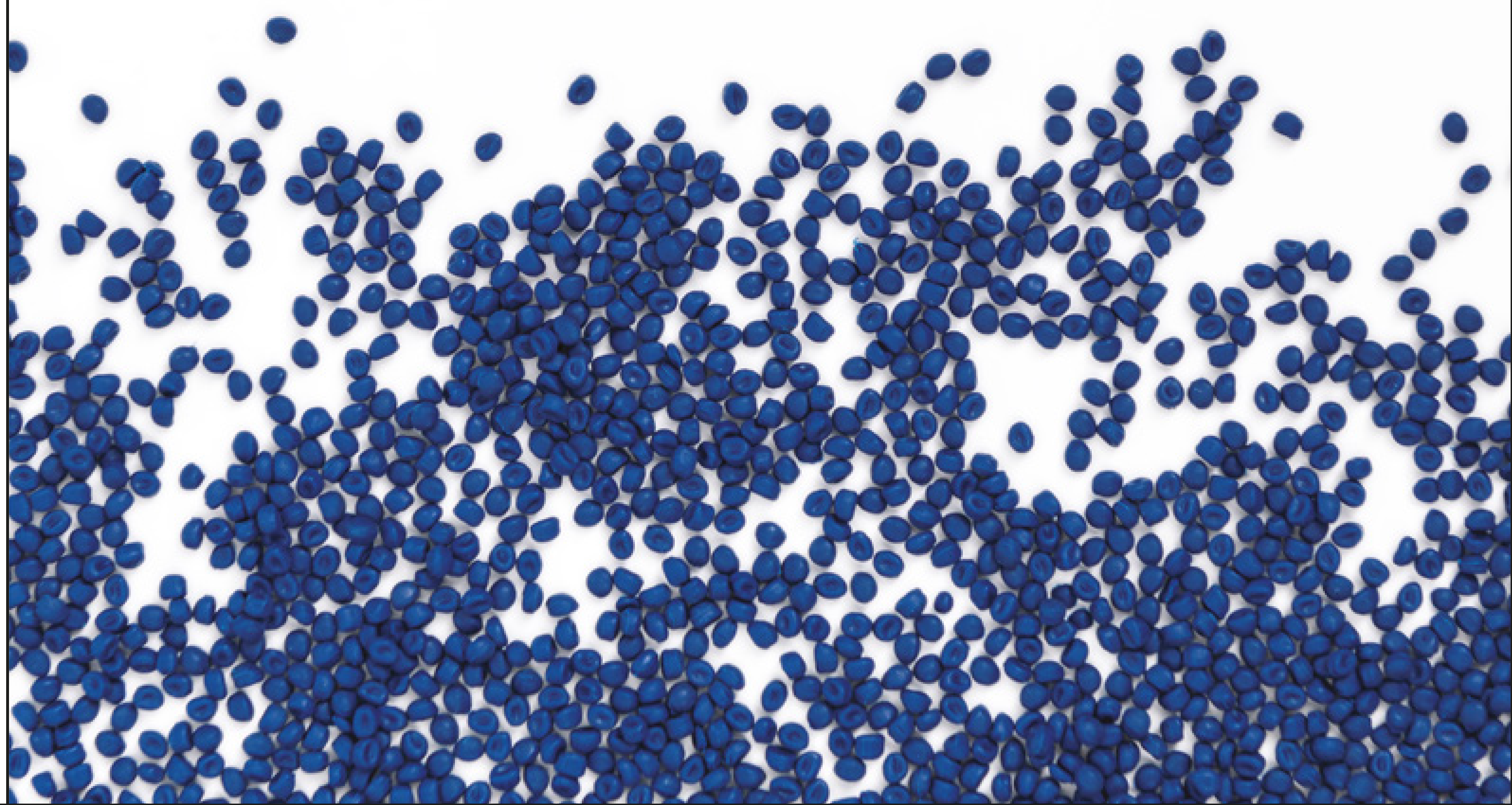
RECOMMENDATION	Elastron D	Elastron G		Elastron V		TPO	FR 6 series		Bondable
		G201	Others	V101	V201		V601	G601	G500
Drying temperatures	No need	90°C	90°C	90°C	90°C	No need	90°C	90°C	80°C
Drying time		2 hours	2 hours recommended	2 hours	2 hours		2 hours	2 hours	3 hours
Rear Zone temp. °C	140 - 150	160 - 190	145 - 175	155 - 175	155 - 175	155 - 175	155 - 175	145 - 175	180 - 200
Middle Zone temp. °C	145 - 160	170 - 200	155 - 185	165 - 185	165 - 185	165 - 185	165 - 185	155 - 185	190 - 210
Front Zone temp. °C	150 - 165	175 - 205	160 - 190	170 - 190	170 - 190	175 - 195	170 - 190	160 - 190	205 - 220
Nozzle Temperature °C	165 - 185	190 - 220	175 - 205	180 - 210	180 - 210	195 - 225	180 - 210	175 - 205	220 - 230
Injection Speed	Low	Mod / High	Low / Mod	Moderate	High	Mod / High	Moderate	Low / Mod	Mod / High
Injection Time sn	3 - 5	1 - 3	3 - 5	2 - 4	1 - 3	1 - 3	2 - 4	2 - 4	1 - 4
Injection Pressure bar	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40	10 - 40
Hold Pressure bar	5 - 20	5 - 20	5 - 20	5 - 20	5 - 20	5 - 20	5 - 20	5 - 20	5 - 20
Back Pressure bar	5- 40	5- 40	5- 40	5- 40	5- 40	5- 40	5- 40	5- 40	5- 40
Screw Speed (rpm)	50 - 200	50 - 200	50 - 200	50 - 200	50 - 200	50 - 200	50 - 200	50 - 200	50 - 200
Mold Temperature °C	25 - 50	25 - 50	25 - 50	25 - 50	25 - 50	25 - 50	25 - 50	25 - 50	25 - 50
Screw Comp. ratio	1.5:1 – 2.0:1	1.5:1 – 3.0:1	1.5:1 – 2.0:1	1.5:1 – 2.0:1	2.0:1 – 4.0:1	2.0:1 – 4.0:1	1.5:1 – 3.0:1	1.5:1 – 3.0:1	2.0:1 – 4.0:1
Screw L/D	18 - 24	18 - 24	18 - 24	18 - 24	18 - 24	18 - 24	18 - 24	18 - 24	18 - 24
Residence time	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot	1 - 2 shot
Cushion size	8 mm	8 mm	8 mm	8 mm	8 mm	8 mm	8 mm	8 mm	8 mm

EXTRUSION REQUIREMENTS

REQUIREMENTS	Elastron D	Elastron G		Elastron V		TPO	FR 6 series	
		G201	Others	V101	V201		V601	G601
Drying temperatures	No need	90°C	90°C	90°C	90°C	No need	90°C	90°C
Drying time		2 hours	2 hours recommended	2 hours	2 hours		2 hours	2 hours
Screw Comp. ratio	1.5:1 – 2.0:1	1.5:1 – 3.0:1	1.5:1 – 2.0:1	1.5:1 – 2.0:1	2.0:1 – 4.0:1	2.0:1 – 4.0:1	1.5:1 – 3.0:1	1.5:1 – 3.0:1
Screw L/D	18 - 30	18 - 30	18 - 30	18 - 30	18 - 30	18 - 30	18 - 30	18 - 30
Feed Zone temp.	140 - 160	165 - 185	150 - 170	155 - 165	155 - 165	160 - 180	155 - 165	150 - 170
Rear Zone temp.	140 - 160	170 - 190	155 - 175	160 - 180	160 - 180	165 - 185	160 - 180	155 - 175
Center Zone temp.	145 - 165	180 - 200	165 - 185	165 - 185	165 - 185	170 - 190	165 - 185	165 - 185
Front Zone temp.	155 - 175	190 - 220	175 - 205	170 - 190	170 - 190	185 - 205	170 - 190	175 - 205
Head temp	155 - 185	195 - 225	180 - 210	180 - 210	180 - 210	190 - 220	180 - 210	180 - 210
Die temp.	165 - 195	205 - 225	190 - 210	185 - 215	185 - 215	195 - 225	185 - 215	190 - 210

Your TPE Solutions Partner!

ENGI-
NEERING
LIFE



ELASTRON WORLDWIDE

+49 DISTRIBUTORS