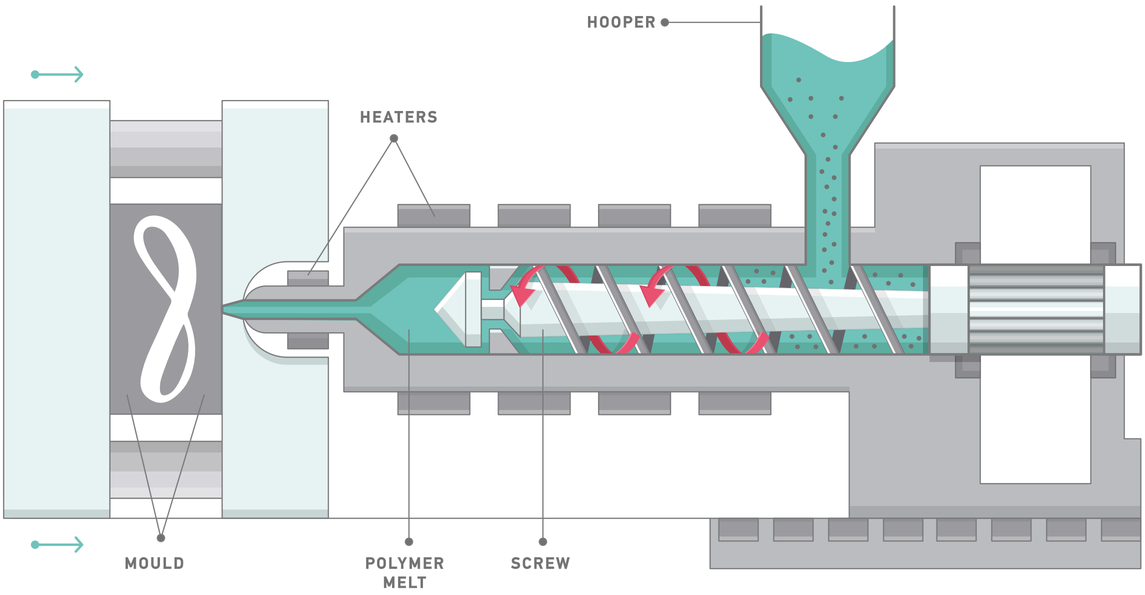
ELASTRON PROCESSING GUIDE FOR INJECTION

THERMOPLASTIC ELASTOMERS

elastron

INJECTION MOLDING

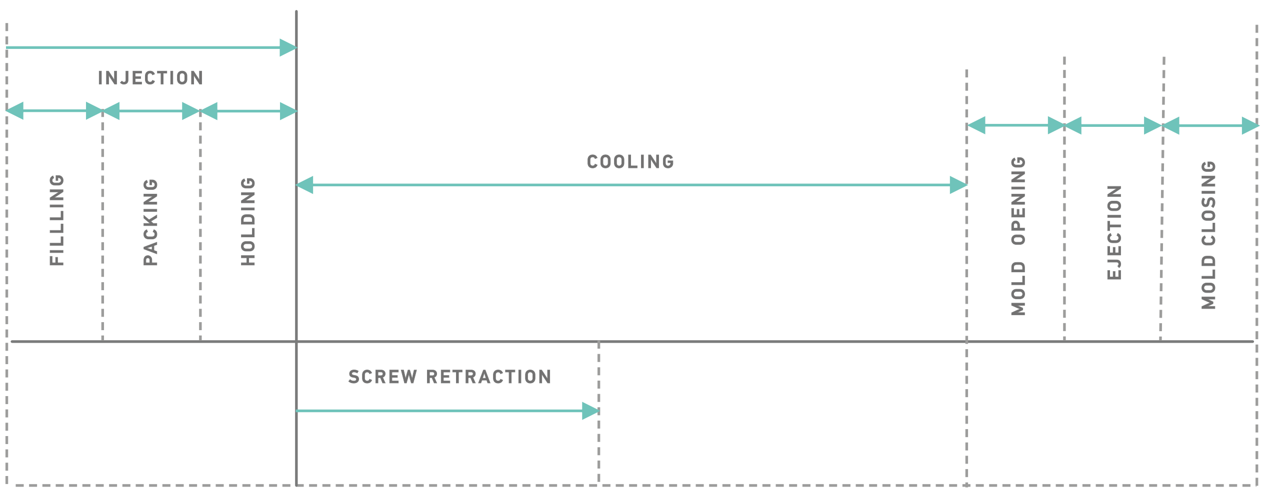


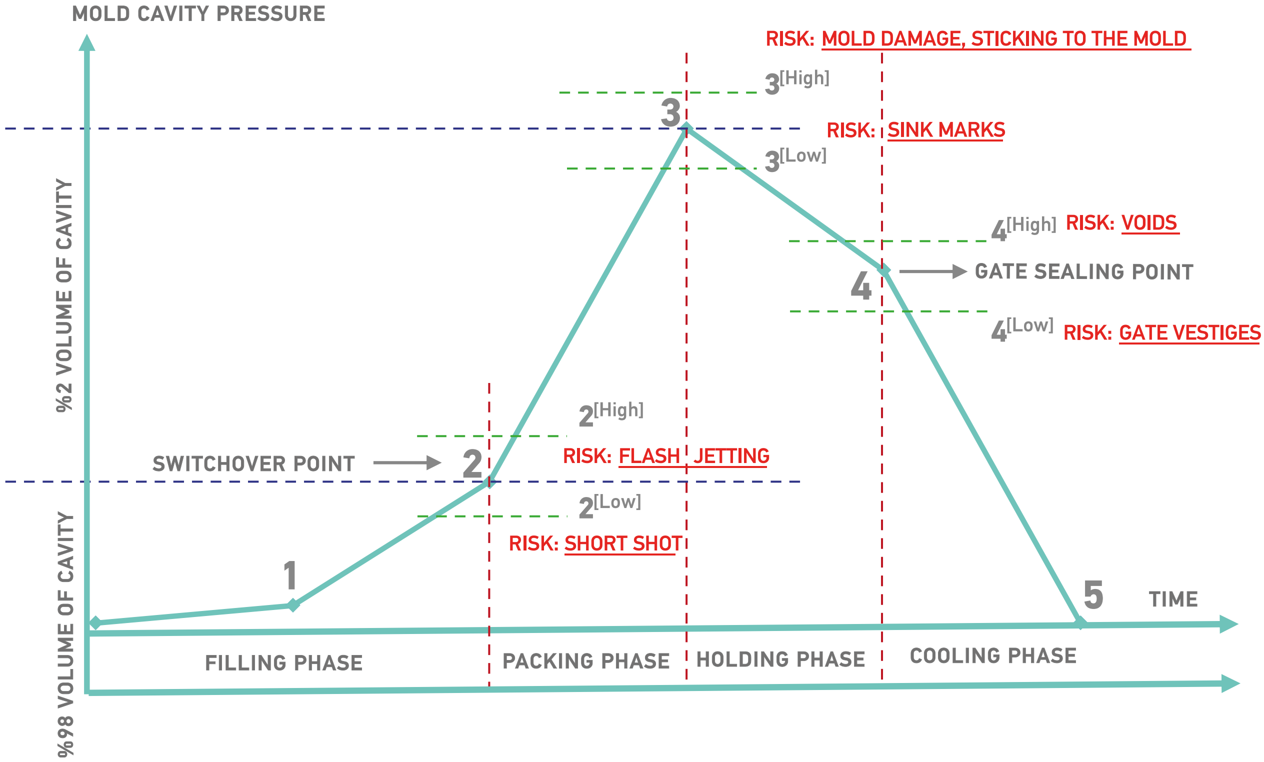
Injection molding machine is consist of two side, a clamping side and an injection side. The main purpose of the clamping side to open and close the mold, and the ejection of products. The purpose of the injection unit are to melt the material by heat being specified and then to inject molten material into a mold.

The screw is rotated with the rpm being 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 with screw retraction, injection phases are stared.

As molten material is introducing in a mold, the injection machine controls the ram speed of the screw, or injection speed (speed profile). In addition to this, it controls molding pressure after molten material fills out cavities.

INJECTION MOLDING CYCLE





1. Filling Phase: Melt must be filled to the mold as fast as possible according to the part geometry. If there are any restricted area having affected to the filling properties, proper speed profile must be determined. Flow characteristics are determined by melt temperature, speed, and shear rate. Too high injection speed can create excessive shear and result in issue such as splay and jetting. High speed filling is terminated at **Switchover point** and low speed packing phase commences.

1.1. Switchover Point: It's 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 is also leads to short shot and longer cycle times.

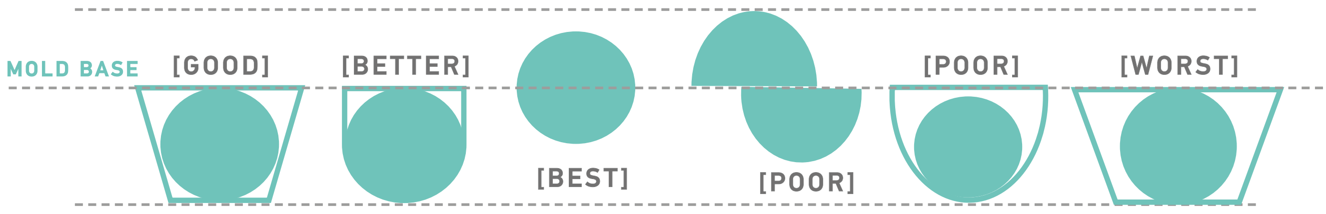
2. Packing Phase: As soon as the material gets into the cavity, cooling starts and it induces shrinkage. That's why, there is a need to inject more material in, in order to prevent this shrinkage. After being loaded %98 volume of cavity, injection speed and pressure must be reduced during this phase. Due to the fact that packing pressure determines part weight and part dimensions, it is important to completely fill the mold avoiding over packing 3[High] or under packing 3[Low].

3. Holding Phase: By reducing the pressure and speed being applied, certain amount of pressure must be applied until gate completely freezing off. Gate freezing is the solidification of material around the gate area and prevents melt backflows out of the mold and let us keep the dimension and weight values stable.

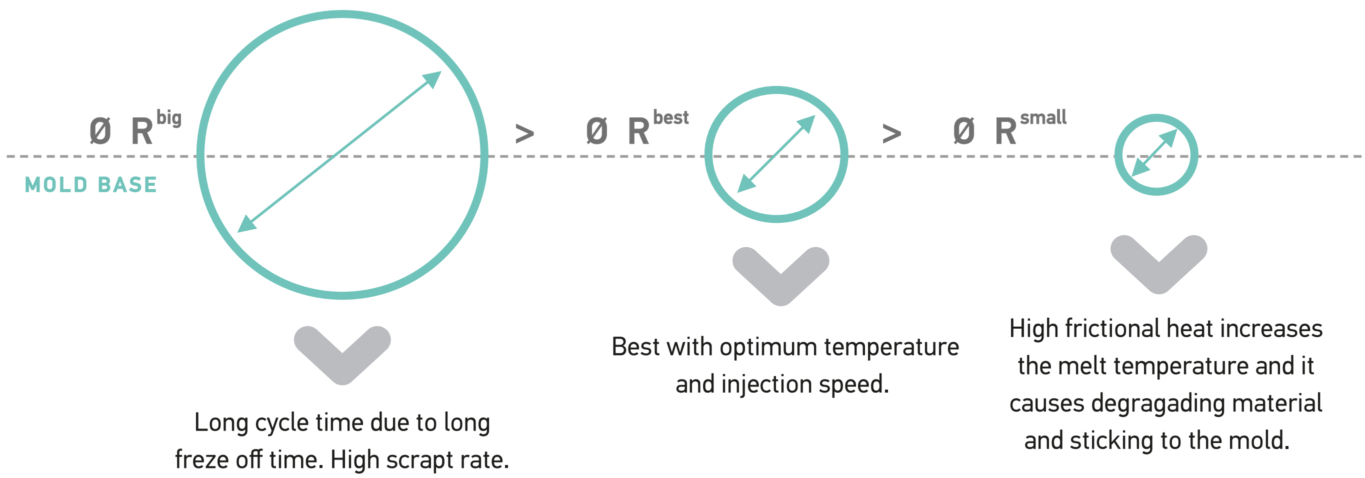
4. Gate Sealing Point: The plastic enters the cavity through the gate. As long as the gate is not frozen, new material comes in to the mold replacing unoccupied free spaces left from shrinking material. At the end of the solidification, gate is completely blocked due to cooled down and no any material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

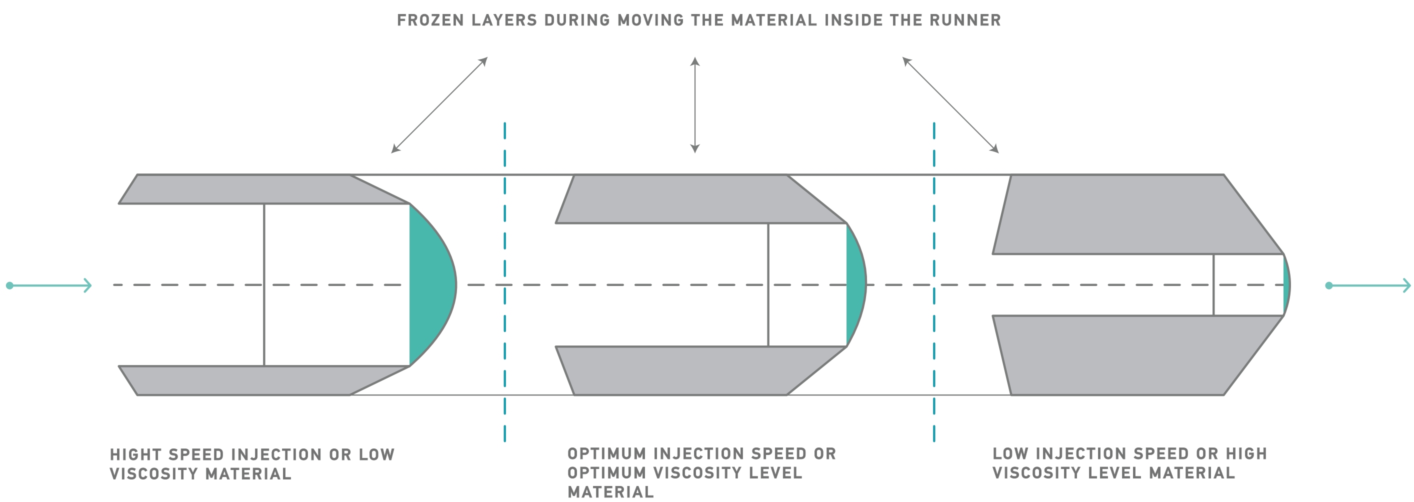
5. Cooling Phase: To ensure optimum molding cycles while maintaining part surface requirements and mold filling capability, a stable mold temperature should be determined and maintained. The cooling time takes about 80% of injection molding cycle, so a well-designed cooling system can shorten the molding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

MATERIAL MOVING IN THE RUNNER



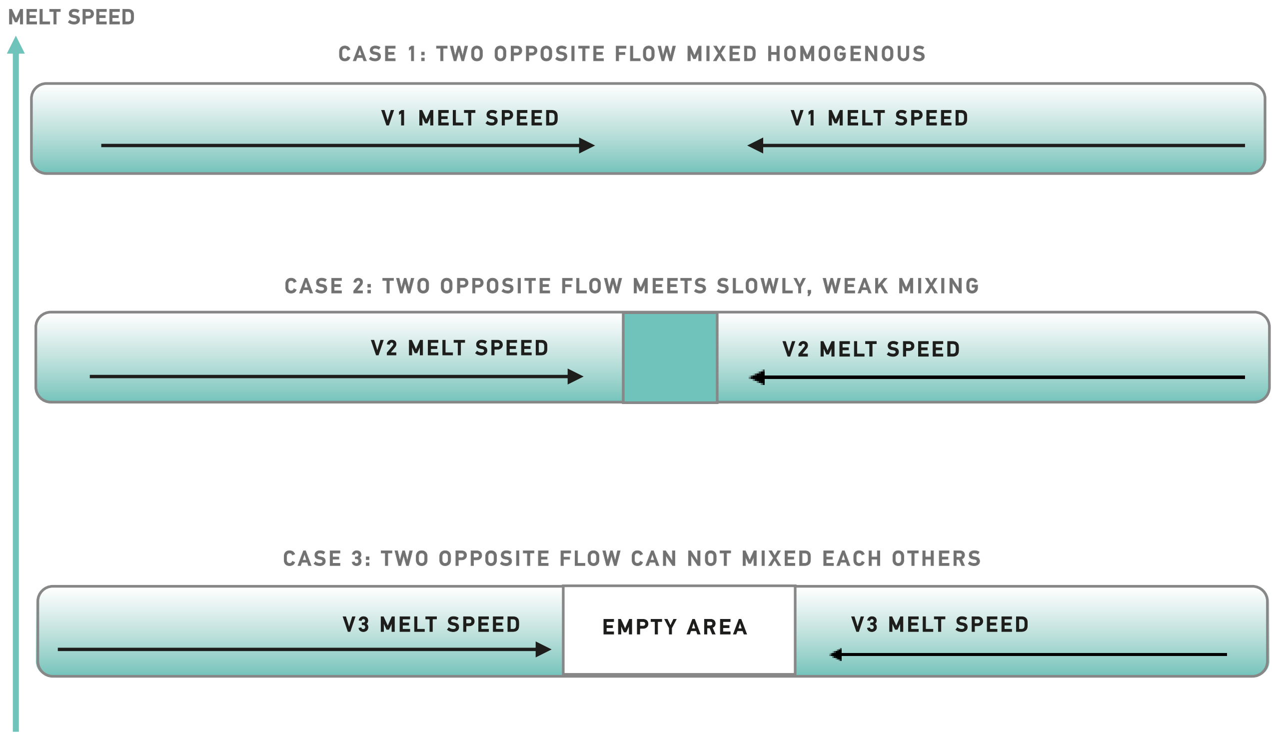
As you see the shapes above the best selection on the runner is fully rounded and centered to the mold base.





During injection, injection speed and material viscosity are two of the most important factors to effect the molding quality. If your material is low viscous material do not use excessive injection speed because high speed generates high shear rates and material gets warmed on the way of runner and it's temperatures exceed the degradation level. In that case your material would get sticked to the cavity wall after injection. If your material is sticking to the mold, reduce your injection speed gradually.

MATERIAL MOVING IN THE MOLD



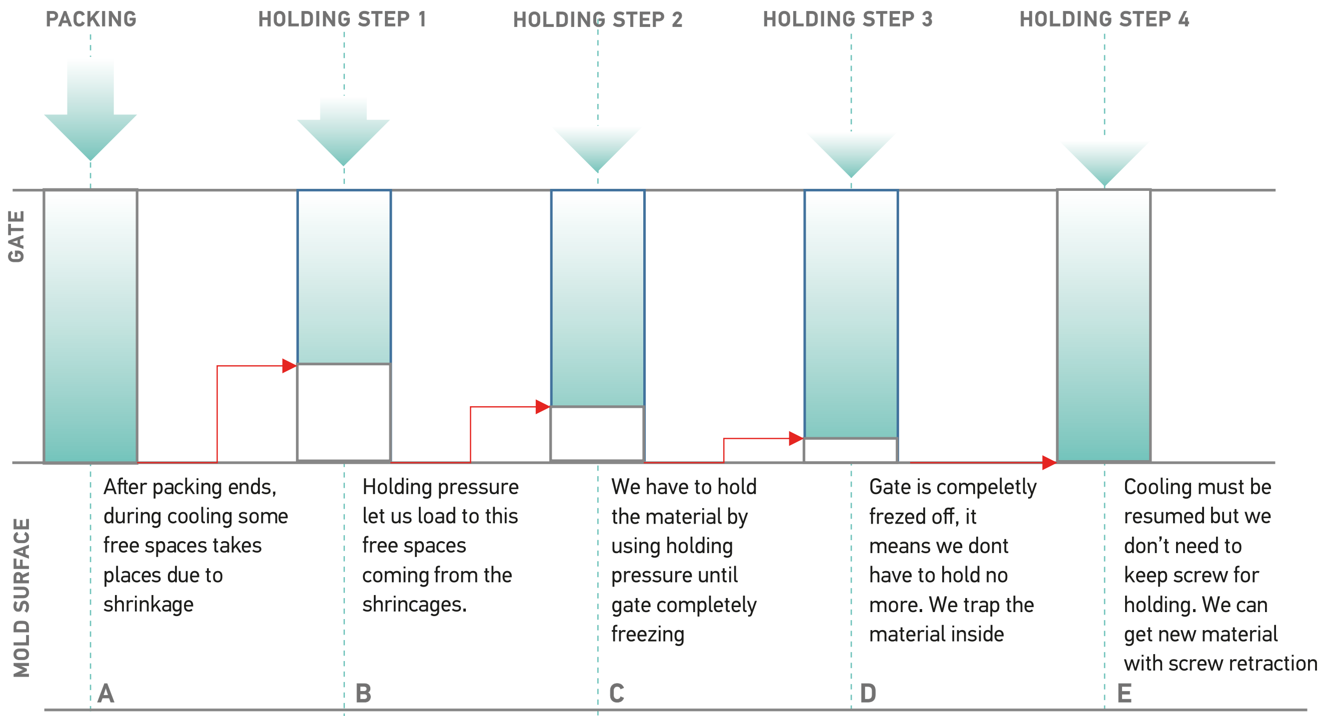
Case 1: Thanks to optimum melt speed, two opposite flow channel meet seamlessly without any weak areas.

Case 2: The same material with low melt speed causes weak mixing. **Cold Weld Line**

Case 3: The same material with very low melt speed causes empty area due to not facing together. **Short Shot**.

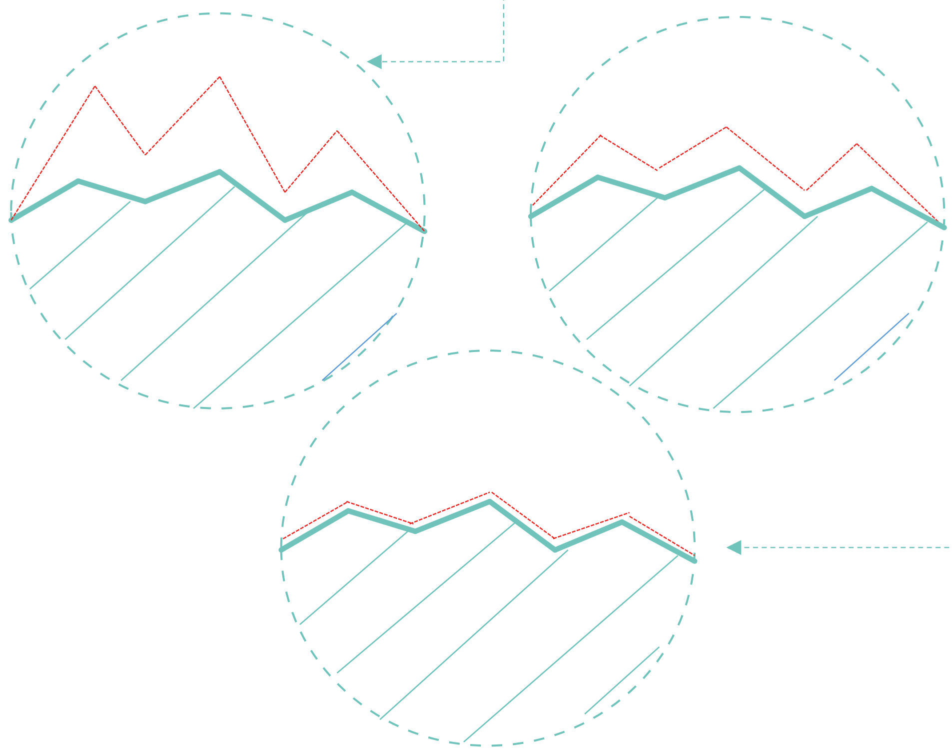
Why Melt Speed is Low

- **Low injection speed:**
  During filling phase, while injection to the cavity through flowpath, melt gets cooled over the time, that's why we must inject the material as fast as possible, otherwise melt would get cooled on the way and there would not be homogenous mixing at the intersection line of two separate flow.
- **High viscous material:**
  Material viscosity might be higher than normal. In this case, material runs slowly in the flow path despite the injection speed being increased.
- **Low mold temperature**
  If mold temperatures are lower than normal, material cannot move in the mold by its natural flow properties, it's also tend to make wavely surface on the part.
- **Unproper gate location**
  Gate
- **Low gate diameter:**
  Means pressure loss for injection speed in the cavity
- **Insufficient melt temperature:**
  Lower melt temp means higer viscosity.



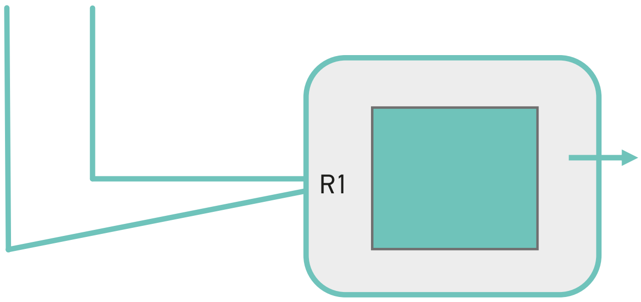
If we don't apply holding or insufficient holding leads

- Sink Marks
- Short Shot
- Surface gloss difference

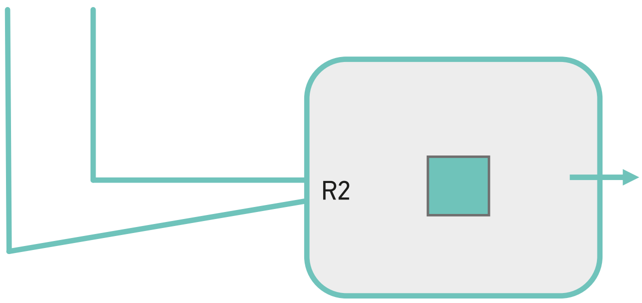


WHY GATE DIAMETER IS IMPORTANT DURING GATE SEALING

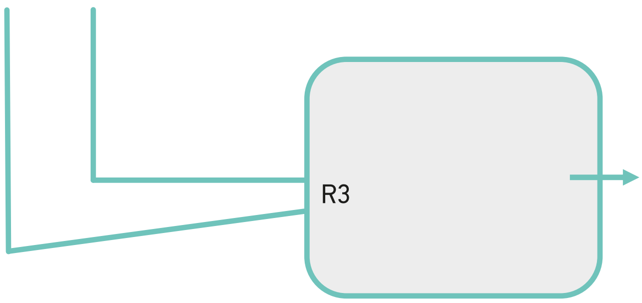
We have to hold the material after packing phase until gate is completely freezing off. Otherwise our liquid material goes back to runner inside over the gate. That's why, gate diameter is so important, because we have to freeze that area. Larger gate diameter increases the holding time, lower gate diameter takes place a premature freezing.



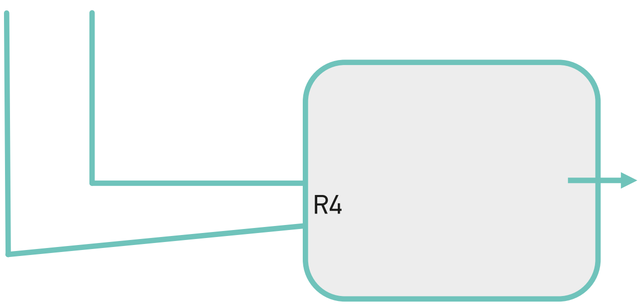
Gater diameter is lower than normal. That means our material is still enough hot even if we have completely sealed the gate. Because due to lower diameter, our gate had frozen earlier than normal but we need to push some material when hot material cools down. In that case, after injecttion, there would be some voids inside at the same time some areas would collapse.



Gater diameter is still not enough diameter beacuse there are some areas need to be cooled. We must increase the diameter gradually.

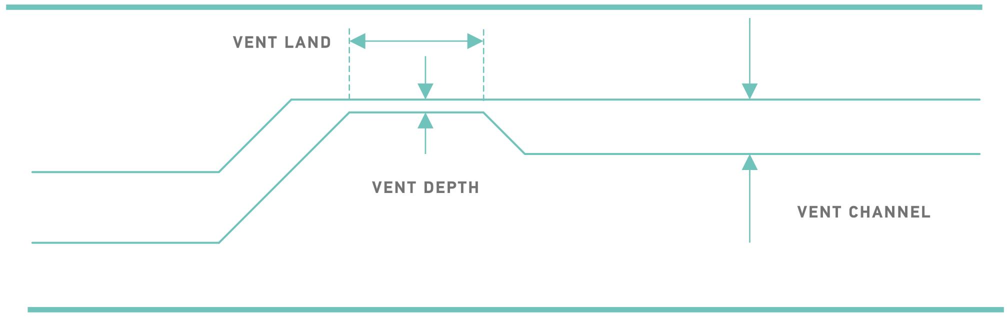


We have reached the best diameter. Due to this we would not see any warpage because after gate sealing there are no any hot areas containing high risk



Higher gate diameter increases gate sealing time so our cycle time would be greater.

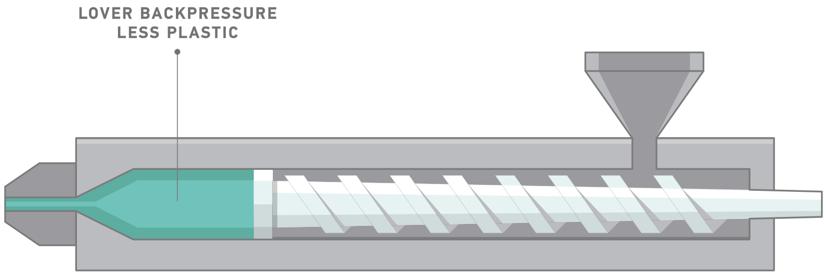
VENT DESIGN

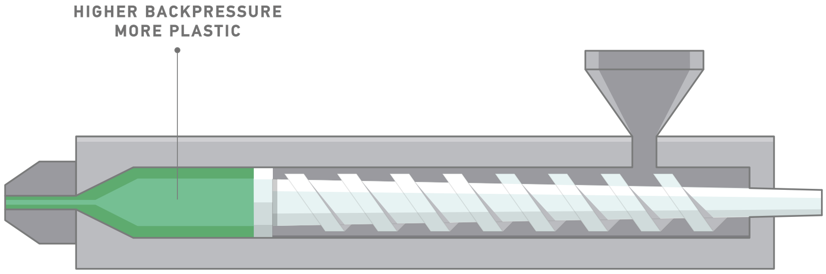


Vent Depth > Normal → Flash, Material sticks to the mold

Vent Channel < Normal → Gas cannot be removed completely.

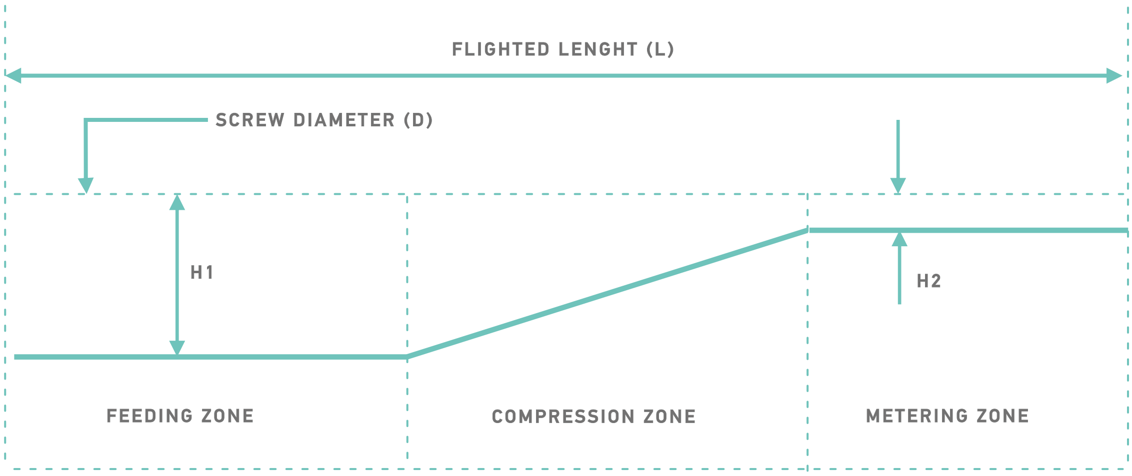
WHY BACK PRESSURE IS IMPORTANT





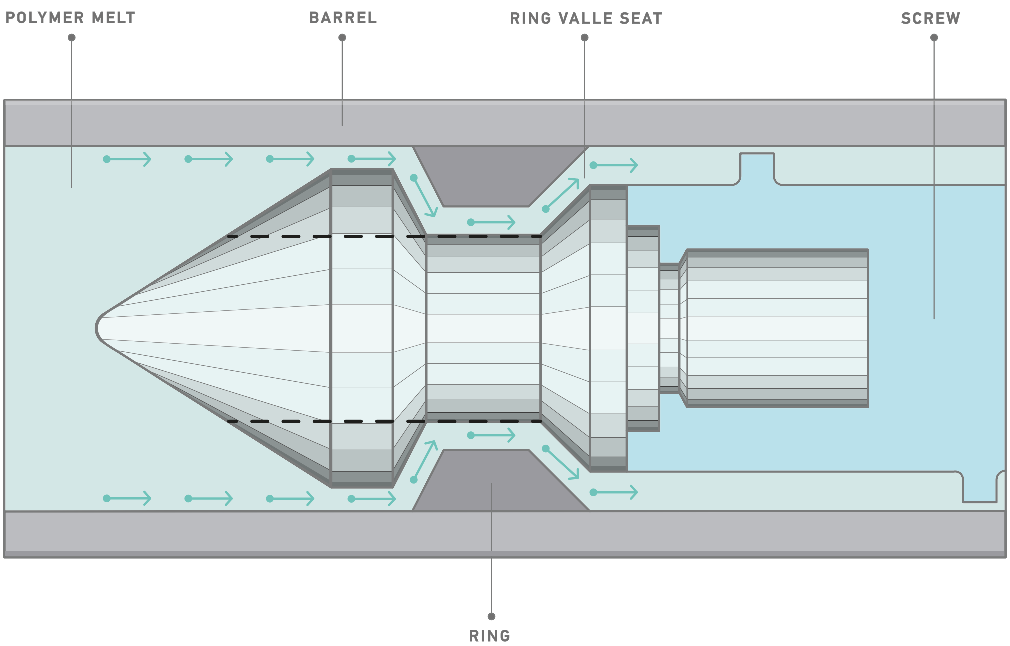
Back pressure is the pressure when taking place on front of the screw during screw retraction to prepare the next shot. Increasing it’s values, collects more material on front of the screw, that’s why if you don’t change the injection speed strokes, your part weight would be heavier than before because of higher density taking place of the same volume in front of the screw.

SCREW



Compression ratio = H1/H2

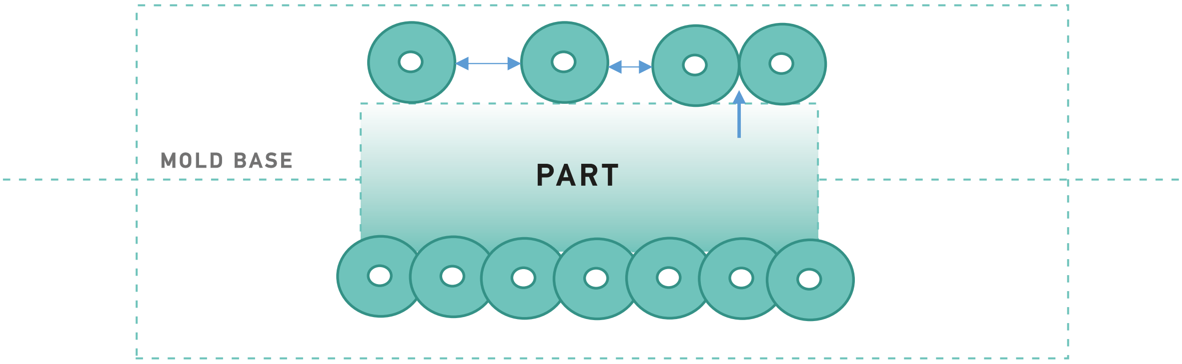
L/D = Flighted Length / Screw Diameter



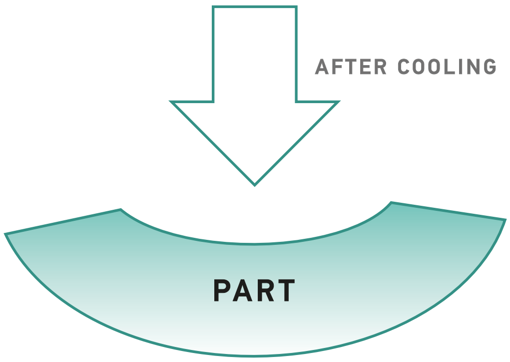
Reciprocating screw injection molding machines equipped with a general purpose screw design are adequate for processing the material.

This screw design, should have 50% of its length as the feed zone, 25% as the compression zone, and 25% as 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 be of the free-flow design and as short as possible.

COOLING



Cooling channels must be installed correctly, if there are any voids between them, cooling effecting the part would not be balanced. That's why warpage issue risk takes place.



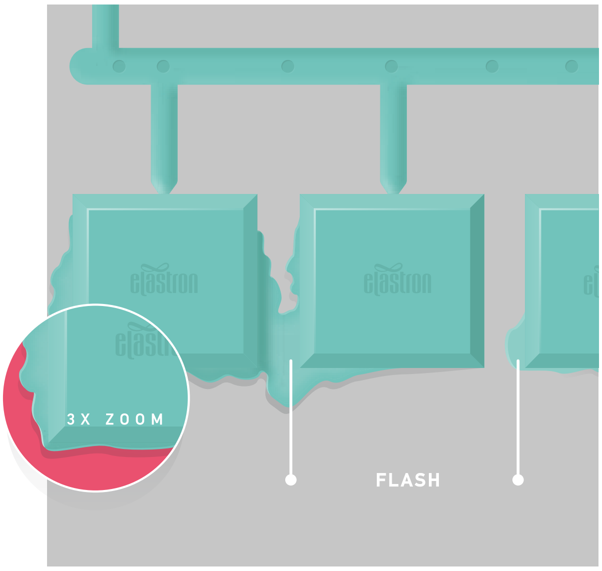
TROUBLESHOOTING

1. Short Shot

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

2. Flash

- Low clamp force: Clamping force must be higher than injeciton 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 flash or foreign material stuck on the parting surface.
- Injection parameters: High melt temperature (lowered the viscosity of the material) or high injection pressure win the calmping force and flash takes place.
- Nonconforming venting: Due to venting design and a very poor venting system, or venting dimension is not properly.

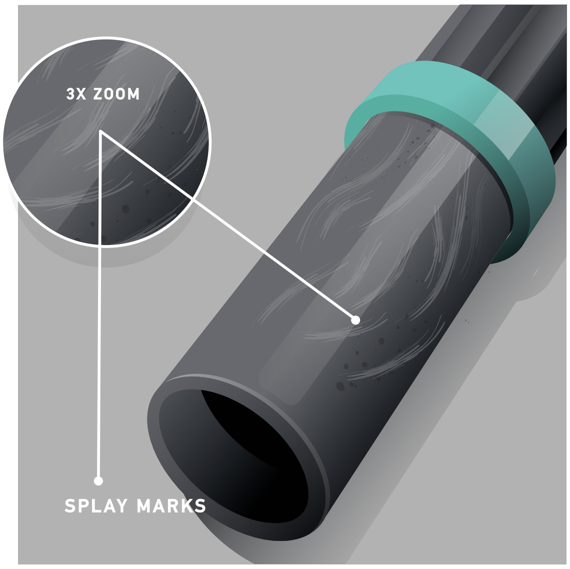


3. Sink Marks

- After injection, holding must be applied until gate completely freezing off, otherwise melt goes back to runner inside and sink marks takes place.
- Short cooling time, because after stress relaxation in cooling, new material loaded to the cavity replacing the voids.
- High melt temperature or mold temperature
- Due to mold design, there might be some pressure loss in order to hold the material after injection.

4. Splay Marks

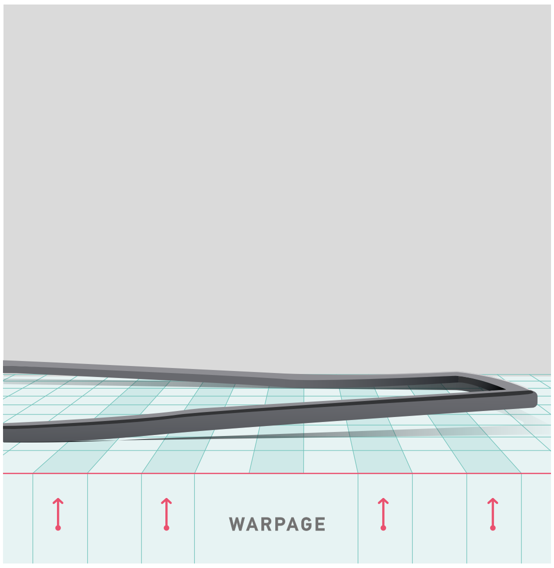
If material are used without predrying, the moisture inside the material can cause this marks. Also high rpm during screw retraction, generates gas and this gas are injected the parts with melted material.



- Dry plastic according to our suggestion.
- Reduce injection speed, because high speed generates gas due to friction.
- Reduce melt temperatures by reducing the barrel temepratures.
- Reduce screw rpm, high rpm generates gas.
- Increase back pressure, ıncreasing the back pressure discharge the gas through the hooper.
- Increase mold temperatures
- Increase venting
- Increase gate diameters, because tight diameter generates gas due to high shear rates.

5. Warpage

A differential cooling rate of the melt in two sections of the molded product.

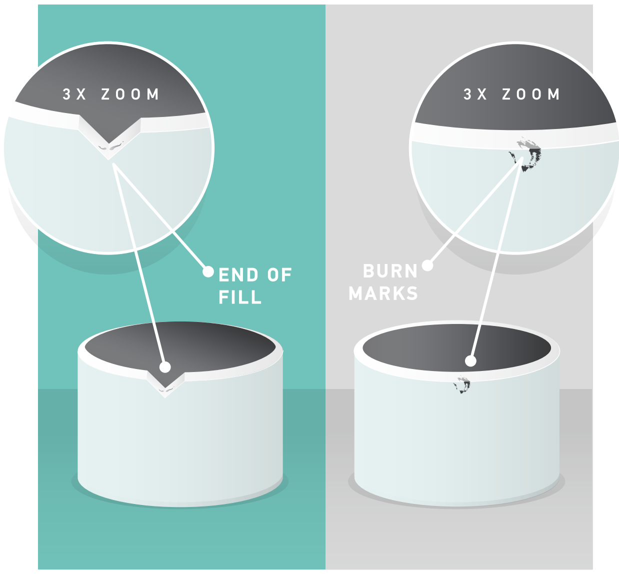


- Reduce melt temperature to reduce the cooling time.
- Reduce the mold temperature to reasonable level according to our requirements
- Increase pack and hold pressures, if any pressure loss.
- Increase pack and hold times,
- Due to unsufficeient cooling time warpage takes place after ejection. That's why increase the cooling time to cool the part to suffient level.

6. Burn Marks

Material degradation

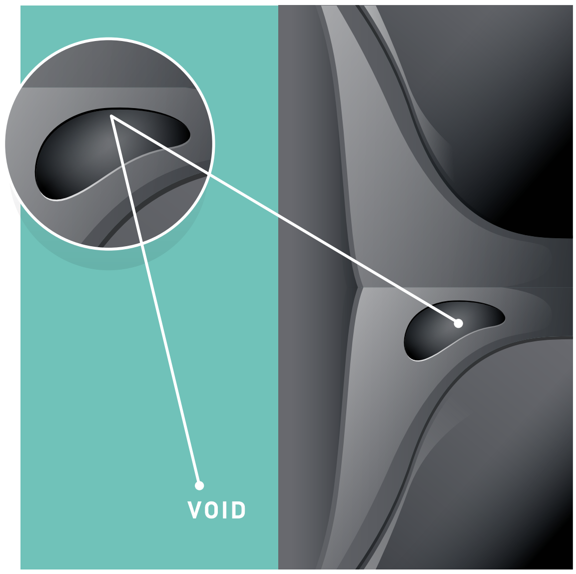
Burn marks can result from the degraded materials and then appears on the surface of the molded part or near the venting areas. Material degradation is caused by:



- High melt temperature: Excessive melt temperature can be caused by higher barrel temperature setting, or malfunctioning in the temperature controller.
- High screw rotation speed: If the screw rpm is too high during the plasticization time during screw retraction, that will create too much frictional heat, which could degrade the material.
- Restrictive flow path: When the melt flows through the flowpath which icludes restrictive nozzle, runner, gate, or part sections, shear rates gets incresedheat, which could degrade the material.
- High speed injection: Creates extra temperatures due to frictional heat in the runner.

7. Voids

Occurs 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 sucks a vacuum void on the inside of the part

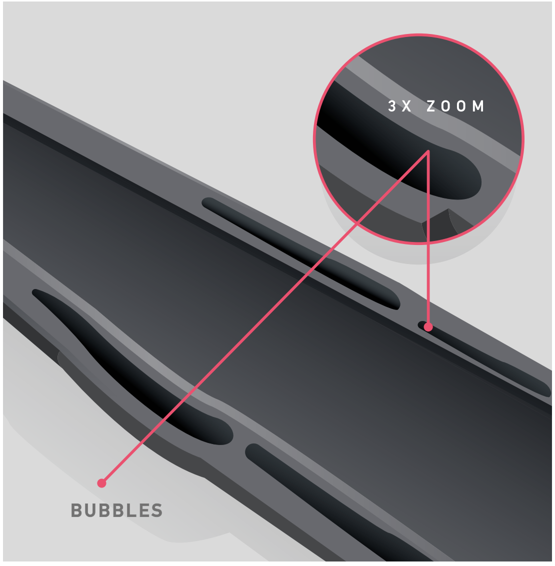


- Reduce melt temperature: Cooling after high temperature might cause the voids.
- Reduce mold temperature: High mold temperatures keeps the melt temperatures high, that’s why thiskind of voids occur.
- Reduce injection speed: High speed injection creates high frictional shear and melt cooling would 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

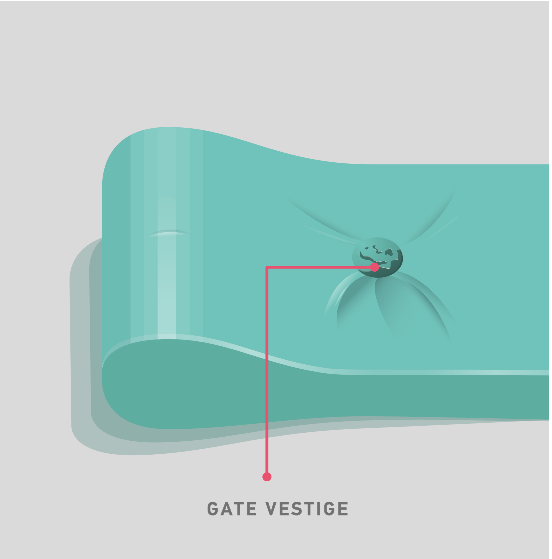
Moisture or gase coming from the product gets injected in the mold cavity this moisture or gas if embedded inside the melt can show up as bubbles.

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



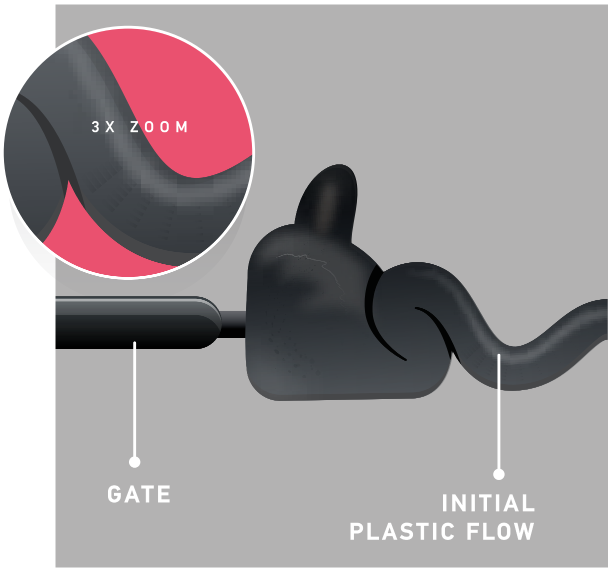
9. Gate Vestige

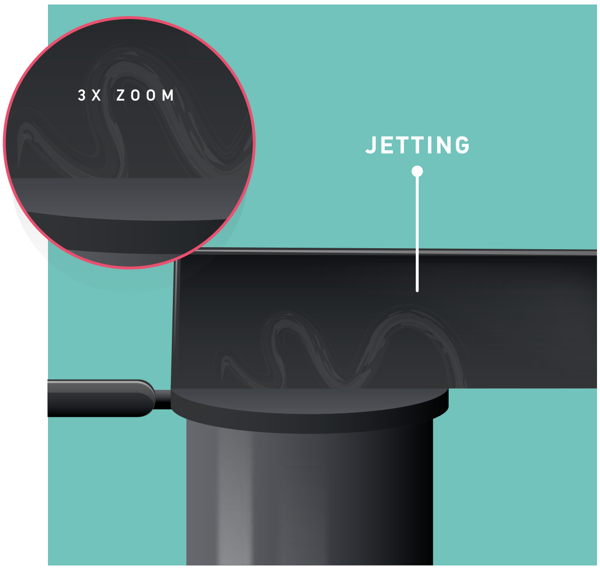
- When holding phase, reduce the injection speed and pressure
- Arrange the injection speed profile properly,
- Increase the melt temperature by using the barrel temperatures gradually



10. Jetting

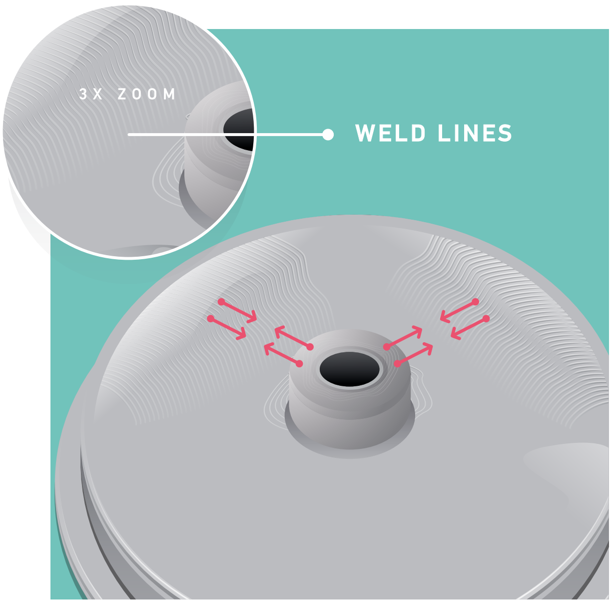
- Place the gate to against to metal surface.
- Use an overlap gate or a submarine gate
- Slow down the melt with a gradually divergent flow area.
- Reduce injection speed
- Increase melt temperature





11. Weld Lines

- Increase melt temperature: If two different flow meets together as colder than normal, such king of traces would be coming out.
- Increase mold temperatures: Cold mold wall prevents the material flow.
- Increase injection speeds: Lower injection speed causes premature cooling.
- Increase venting: Ventilation supply us better flowing in the mold.



PROCESSING GUIDE FOR INJECTION

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 2 hours recommended	90 °C	90 °C	No need	90 °C	90 °C	80 °C
Drying time		2 hours		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	**Other**	**V101**	**V201**		**V601**	**G601**
Drying temperatures	No need	90 °C	90 °C 2 hours recommended	90 °C	90 °C	No need	90 °C	90 °C
Drying time		2 hours		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

elastron

elastron.com
elastron@elastron.com

elastron-tpe.de
info@elastron-tpe.de

elastronusa.com
elastron@elastronusa.com

THERMOPLASTIC ELASTOMERS