Product/Process Knowledge Document

Injection Molding Process

325 E. Eisenhower Parkway, Suite 4

Ann Arbor, MI 48108

(734) 761-4940

Omnex provides training, consulting and software solutions to the international market with offices in the USA, Canada, Mexico, Latin America, Germany, China (PRC), India, the Middle East and Thailand. Omnex offers over 200 training workshops in business, quality, environmental, laboratory, health & safety management systems and Lean Six sigma worldwide.

Internet email: info@omnex.com

Web: www.omnex.com

INTRODUCTION - INJECTION MOLDING

The injection molding has seen a stable growth since its beginnings in the late 1800's. The technique has evolved from the production of combs and buttons to major consumer, industrial, medical, and aerospace products.

Injection molding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide variety of products are manufactured using injection molding, which vary greatly in their size, complexity, and application.

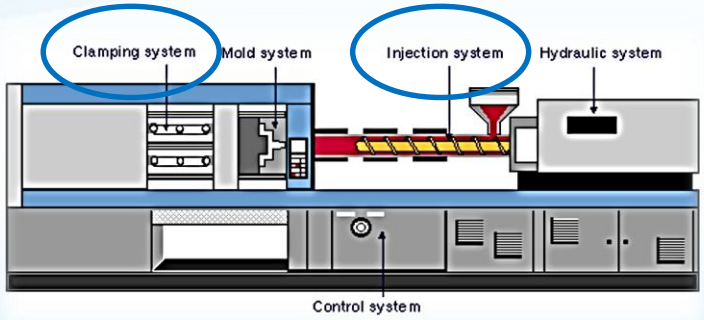
The injection molding process requires the use of an injection molding machine, raw plastic material, and a mold. The plastic is melted in the injection molding machine and then injected into the mold, where it cools and solidifies into the final part.

Technical details

- **<u>The process cycle for injection molding is very short, typically from seconds to few minutes, and consists of the following four stages:</u>**

- *Clamping* - Prior to the injection of the material into the mold, the two halves of the mold must first be securely closed by the clamping unit. Each half of the mold is attached to the injection molding machine and one half is allowed to slide. The hydraulically powered clamping unit pushes the mold halves together and exerts sufficient force to keep the mold securely closed while the material is injected. The time required to close and clamp the mold is dependent upon the machine.

- *Injection* - The raw plastic material, usually in the form of granules, is fed into the injection molding machine, and advanced towards the mold by the injection unit. During this process, the material is melted by heat and pressure. The molten plastic is then injected into the mold very quickly and the buildup of pressure packs and holds the material. The amount of material that is injected is referred to as the shot.



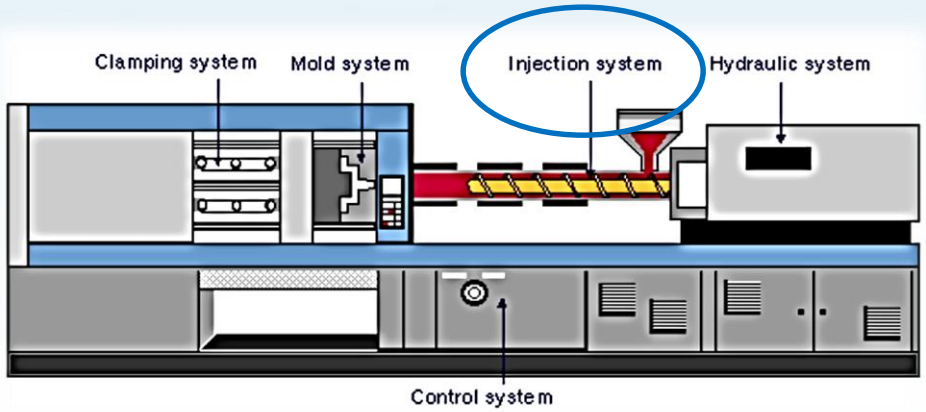
Technical details

- *Cooling* - The molten plastic that is inside the mold begins to cool as soon as it makes contact with the interior mold surfaces. As the plastic cools, it will solidify into the shape of the desired part. However, during cooling some shrinkage of the part may occur. The packing of material in the injection stage allows additional material to flow into the mold and reduce the amount of visible shrinkage. The mold can not be opened until the required cooling time has elapsed.

- *Ejection* - After sufficient time has passed, the cooled part may be ejected from the mold by the ejection system, which is attached to the rear half of the mold. When the mold is opened, a mechanism is used to push the part out of the mold. Force must be applied to eject the part because during cooling the part shrinks and adheres to the mold. In order to facilitate the ejection of the part, a mold release agent can be sprayed onto the surfaces of the mold cavity prior to injection of the material.

Technical details

- *Injection unit* - The injection unit is responsible for both heating and injecting the material into the mold. The first part of this unit is the hopper, a large container into which the raw plastic is poured. The hopper has an open bottom, which allows the material to feed into the barrel. The barrel contains the mechanism for heating and injecting the material into the mold. This mechanism is usually a ram injector or a reciprocating screw.



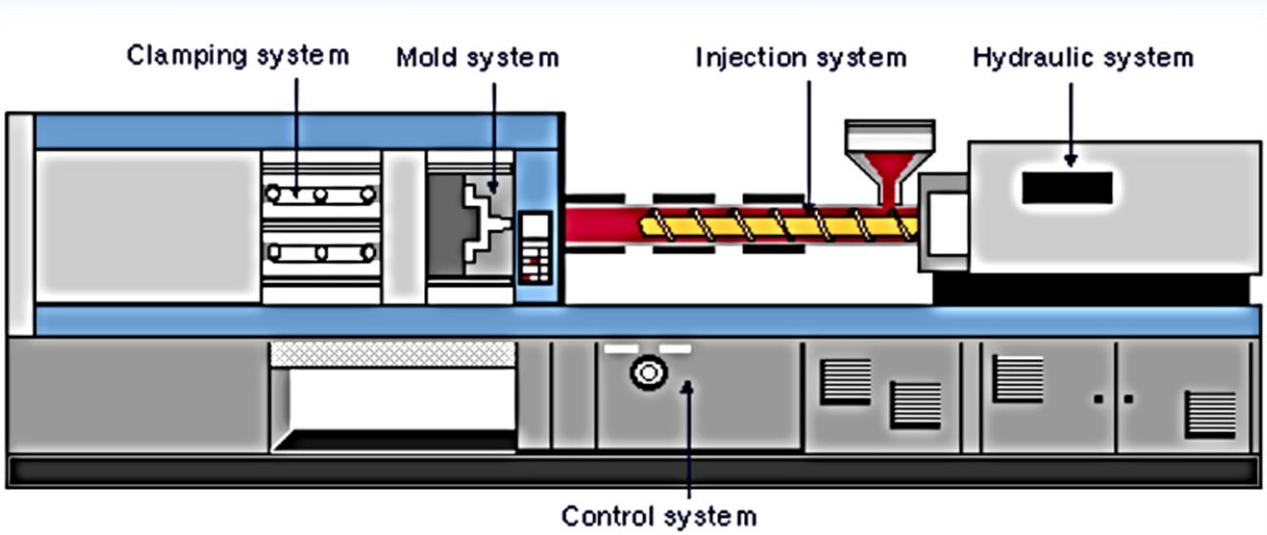
Advantages of Injection Molding

- Thermoplastics are recyclable
- Good design flexibility
- Smaller lead times
- Increased corrosion resistance
- Can convert multiple metal parts into a single plastic part
- Less Post-Production Scrap
- Lower packaging and shipping costs
- Higher strength to weight ratio as compared with metal
- Lighter weight and improved ease of use
- Low manufacturing costs for high production rate

Disadvantages of Injection Molding

- More lead time for mold manufacturing
- High Prototyping Cost
- Difficult to estimate the Cost
- High initial tooling Cost
- Part design restrictions in some cases

WHAT IS INJECTION MOLDING ?



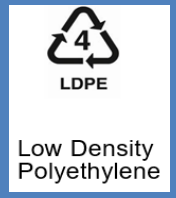
The injection machine is a machine that melt plasticize the molding material inside the heating cylinder and inject this into the mold tool / Cavity to create the molded product by solidifying inside it.

The injection machine is developed with a mold clamping device that opens and closes the mold tool, and device that plasticize and inject the molding material.

Plastics classification

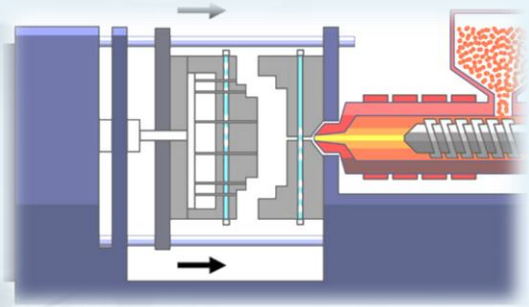
Type of Plastics	Properties	Plastics uses
	Good gas & moisture barrier properties High heat resistance Clear Hard Tough Microwave transparency Solvent resistant	Water, Drink and beer bottles Pre-prepared food trays and roasting bags Boil in the bag food pouches Some shampoo and mouthwash bottles
	Good moisture barrier properties Good chemical resistance Stress resistant	Bottles Snack food boxes Milk and non-carbonated drinks bottles Toys, buckets, rigid pipes, crates, plant pots, garden furniture's etc.
	Excellent transparency Hard & Rigid Good chemical resistance Long term stability Good weathering ability Stable electrical properties	Credit cards Carpet backing and other floor covering Window and door frames, guttering Pipes and fittings, wire and cable insulation, Floors coatings etc.

Plastics classification

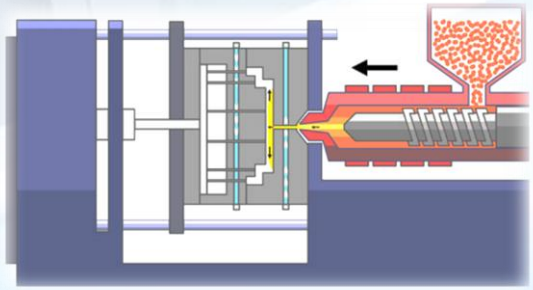
	**Tough and flexible** **Soft – scratches easily Good transparency** **Low melting point** **Stable electrical properties**	**Packaging films, bubble wrap Flexible bottles** **Irrigation pipes** **Thick shopping bags** **Wire and cable applications** **Some bottle tops**
	Excellent chemical resistance High melting point Hard, but flexible Strong	Car Interior plastic parts and bumpers etc Bottle tops Syrup bottles crisp bags, biscuit wrappers Crates, plant pots, drinking straws
	Hard Brittle	Fast food trays Video cases Vending cups and disposable cutlery Seed trays Coat hangers Low cost brittle toys
	There are other polymers that have a wide range of uses, particularly in engineering sectors. They are identified with the number 7 and OTHER	Nylon (PA) Acrylonitrile butadiene styrene (ABS) Polycarbonate (PC) Layered or multi-material mixed polymers

Injection molding process

- **THE BASIC INJECTION MOLDING PROCESS**
- **STEP** 1: The mold closes

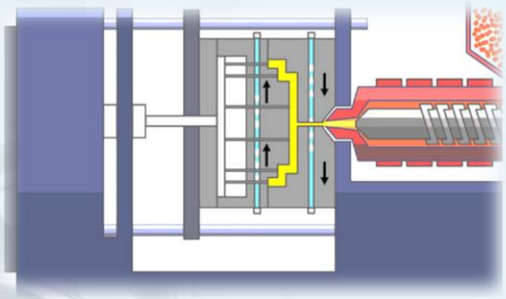


- **STEP** 2: INJECTION
- The heated plastic is injected into the mold. As the heated plastic enters the mold, the displaced air escapes through air vents in the injection pins and along the parting line. Runner, gate and vent design are important to insure the mold is properly filled

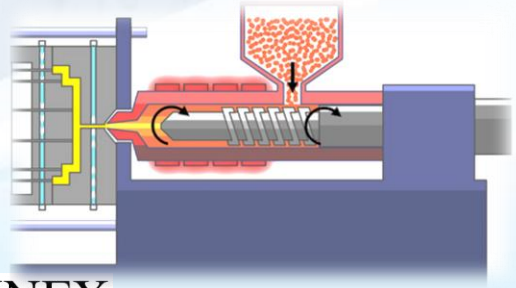


Injection moulding process

- **STEP** 3: COOLING
- Once the mold is filled the part is allowed to cool for the exact amount of cooling time needed to solidify the material. Cooling time is dependent on the type of plastic used and the thickness of the part. Each mold is designed with internal cooling or heating lines where water is cycled through the mold to maintain a constant temperature

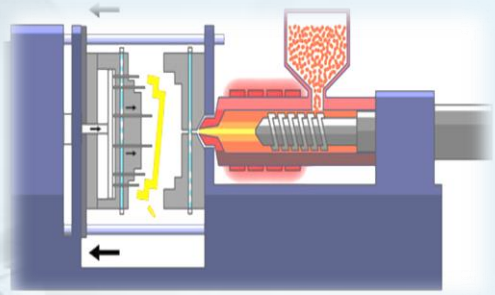


- **STEP** 4: PLASTICIZING THE GRANULES
- While the part cools, the barrel screw retracts and draws new plastic resin into the barrel from the material hopper. The heater bands maintain the needed barrel temperature for the type of resin being used.

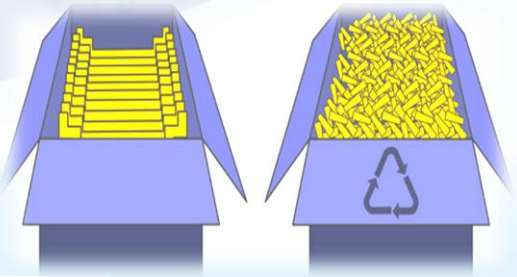


Injection moulding process

- **STEP** 5: EJECTION
- The mold opens and the ejector rod moves the ejector pins forward.
- The part falls and is captured in a bin located below the mold.



- **STEP** 6: REMOVING THE RUNNER AND PACKAGING
- While the part cools, the barrel screw retracts and draws new plastic resin into the barrel from the material hopper. The heater bands maintain the needed barrel temperature for the type of resin being used.

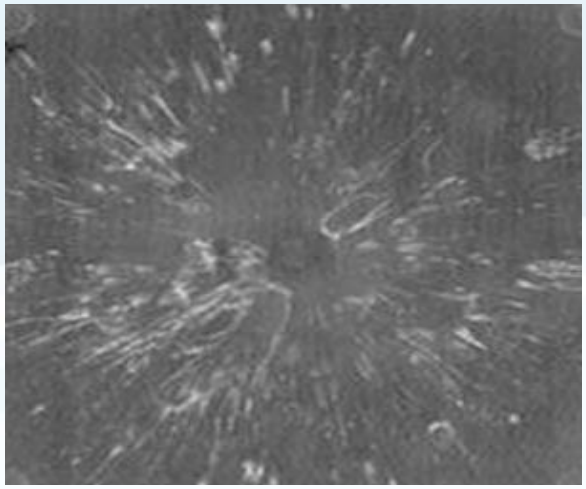


Molding Defects and their causes

Splay Marks / Silver Streaking:

Silver streaks are the splash appearance of moisture, air, or charred plastic particles on the surface of a molded part, which are fanned out in a direction emanating from the gate location

Screw rpm too high.
Back pressure too low.
Melt temperature too high.
Nozzle too hot.
Nozzle too small.
Gates too small.
Insufficient venting
Burr in runner or gate.
Cracked mould.
High moisture.
Granules contaminated.
mould cavity contamination.
High shot size.



Burn Marks

Burn marks are discolorations, usually rust colored, that appear on the surface of the injection molded parts. Burn marks on injected molded parts usually occur near the end of the flow path or where air became trapped

Faulty temperature controllers.

Gates too small

Dead material deposited on screw or nozzle.

Melt stock temperature too high or too low.

Nozzle diameter too small

High Injection speed

High injection pressure.

Inefficient mould temperature.

Improper Venting.

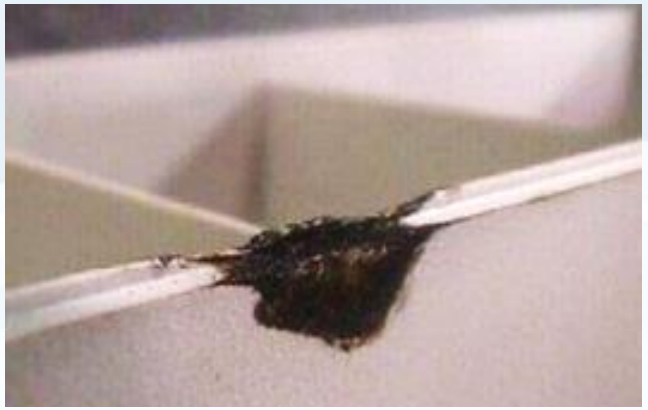
Improper gate location

Front zone temperature too high.

Screw speed too high.

High back pressure.

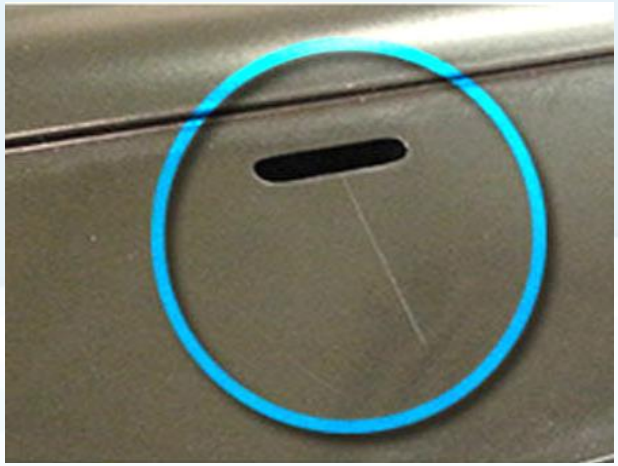
Compression ratio of screw too high.



Weld Lines

Weld line is a line where two flow fronts meet when there is the inability of two or more flow fronts to Joint or weld together during the molding process

Injection speed too slow.
Melt stock temperature to low.
Material too cold.
Contamination in the raw material.
mould temperature to low.
Injection pressure too low.
Insufficient mould venting
Cylinder temperature too low.
Injection back pressure too low.
Nozzle diameter too small.
High screw flights in metering zone.
Distance from gate High.
Ineffective flow pattern.
Inadequate flow.



Bubbles

In thick sections of a part, the center cools slowly and the polymer shrinks more, pulling away from itself to form a bubble.

Causes:

Injection pressure too low

Packing time too short

Insufficient feed of material

mould temperature too low.

Injection speed too high

High cushion

At the side of a rib; rib too thick.

Runners or gate too small or badly positioned.

Flow Lines

Surface defect in which circular ripples or wavelets appear near the gate

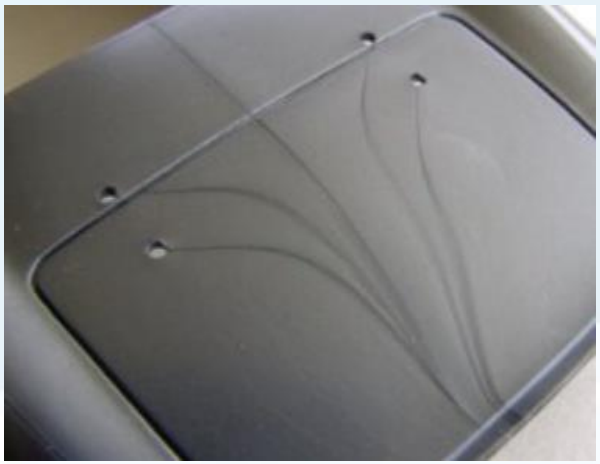
Causes:

mould temperature too low.

Stock temperature too low.

Runners too small

Improper gate size and/or location.



High Warpage/ Shrinkage

High dimensional change in a part after processing, or the High decrease in dimension in a part through cooling.

Short mould close time.
Less cooling time
No chiller or cooling circulation
mould temperature too high
Injection and holding pressure too high or low.
Melt temperature inadequate.
High nozzle and metering zone temperatures.
Too many stresses in part.
Not injected full material in the cavity or mold.
Non-uniform part ejection.

Sink mark

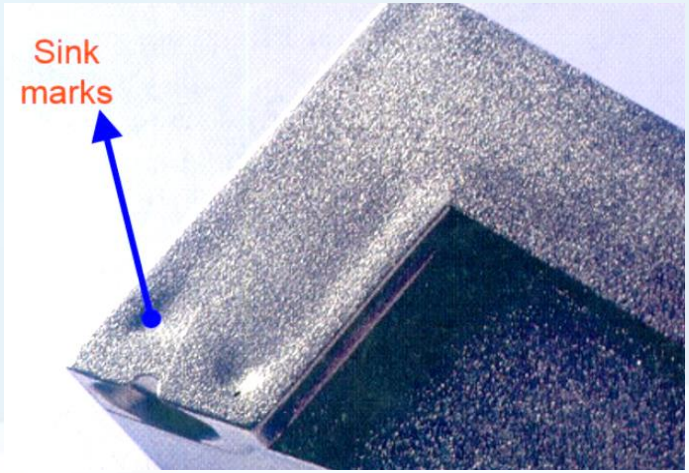
Sink marks are unintended depressions on the surface of a part that do not mimic the mold steel surface

Causes:

Low injection and packing pressure

Short hold time or cooling time

High melt temperature or mold temperature



Flash

Excess plastic around the area of the mould parting line on a moulded part.

mould parting surfaces do not seal properly.

Injection pressure too high.

Clamp pressure set too low or projected area or item too large for clamp pressure of the machine.

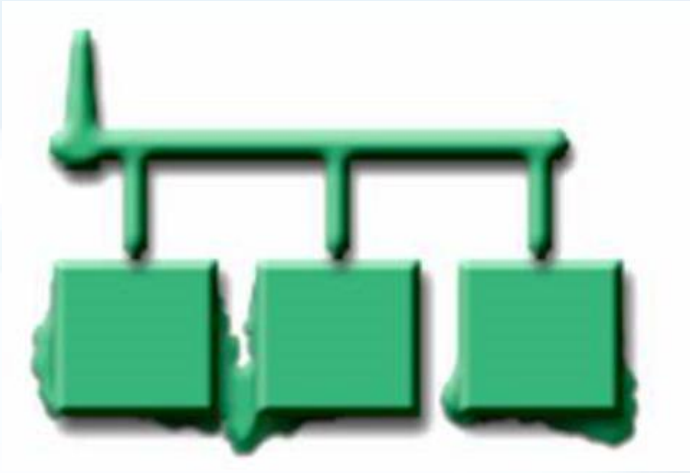
Injection temperature too high.

Feed needs adjustment.

Hold time too long.

Inadequate mould supports.

Oversize vents.



Cracking

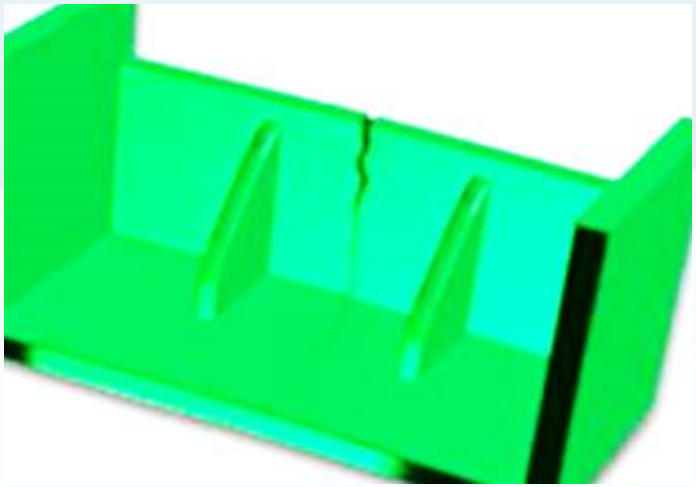
Fracture of the plastic material in an area around a boss, projection, or moulded insert.

Causes:

Parts cool too quickly

moulded-in stress

Wall thickness too heavy for compound.



Low Gloss

Surface roughness resulting from high speed fill which causes surface wrinkling as the polymer melt flows along the wall of the mould.

Causes:

More material injected in the cavity or mold

Inadequate polish of mould surface.

Material or mould too cold.

Air entrapment.

Melt index of material too low.

Improper mould design.

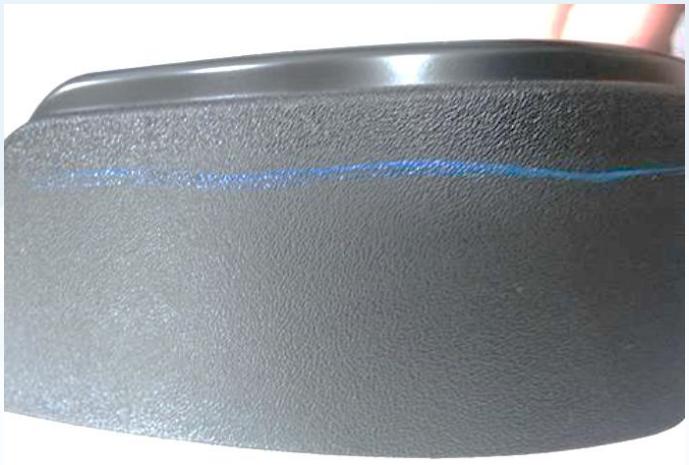
Wrong injection pressure.

High injection speed.

Inadequate flow.

Contamination

Granules Highly moist



Short filling / short mold

Injection of insufficient material to fill the mould.

Causes:

High screw flights in metering zone.

Insufficient venting.

Melt index of resin too low.

Mold temp is very less

Insufficient feed

Less injection pressure.

Less injection speed.

Insufficient booster or injection high-pressure time.

Inefficient screw delay.

Melt temperature too low.

Cylinder temperature inadequate.

mould temperature too low.

Gates or runners too small.

THANK YOU

Info-in@Omnex.com

Are there any Questions?