PENTAGON PLASTICS
moulding Britain since 1972

INJECTION MOULDING - TROUBLE SHOOTING GUIDE

Your trusted partner for
Plastic Injection Moulding

Trouble Shooting Guide - Index

Trouble Shooting Guide for Injection Moulding

Main Cause of Defects

1. The quality of the Melt can be a cause of material deteriation and give inadequate drying
2. Compatibility of polymer additives
3. Quality of the polymer grade for the application process conditions
4. Polymer fillers
5. Process Conditions
6. Maintance of mould or moulding Machine
7. Hot runner system design, installation in the mould and temperature
8. Mould Design, gate position, gate size, venting, ejector position
9. Uneven mould cooling design

Questions

1. Is the defect irregular or is it occurring in every shot?
2. Is the defect always in the same cavity or is it random?
3. Is the defect always located in the same place on the moulding?
4. Is the defect evident on the sprue or cold runner?
5. Does a different batch of resin influence the defect?
6. Does the defect occur on other moulding machines?

Trouble Shooting Guide for Injection Moulding

CONSISTENT DEFECTS

This will usually indicate a problem in the injection nozzle, the hot runner system or shape and design of the cold runner feed and gate.

1. Investigate the moulding process conditions .
2. Investigate the Product Design.

IRREGULAR DEFECTS

Look at compounding factors such as low melt temperature, backpressure, screw speed and retraction can also be important here.

1. Defects covering a large area.
2. Often already visible on the sprue.
3. Check for decomposition of the polymer melt.

Trouble Shooting Guide for Injection Moulding

DISTORTION

Distortion is perhaps the prevalent problem for the moulding industry today and is normally caused by differences in cooling, shrinkage or orientation from one area of the part to another.

Distortion concentrated in the corners is often due to poor cooling, parts incorporating a variable wall thickness profile can exhibit distortion due to differential shrinkage.

As a general rule crystalline materials usually exhibit relatively high shrinkage and are more susceptible to distortion particularly where variable mould surface temperatures and or variations in wall thickness occur.

Fibre filled compounds often show distortion and reinforced crystalline grades tend to show a greater level of distortion than amorphous grades due to higher natural shrinkage of the base polymer.

Check shrinkage due to differential fibre orientation.

Solutions:

1. Aim for uniform packing of the mould.
2. Experiment with different mould temperatures.
3. Ensure balanced filling of the mould.
4. Increase the injection speed.
5. Ensure uniform and symmetrical cooling of the moulding.
6. Choose a more free-flowing plastic material.
7. Consider changing to a material with lower shrinkage.
8. Reduce differences in wall

See Appendix 1 for Material
Selection Guide and a list of Crystalline & Amorphous materials.

Trouble Shooting Guide for Injection Moulding

FLASH

Flash is a small burr formed around the parting line of the moulding and is normally formed when the mould opens slightly during the filling or holding phase.

Flash can also form on a new mould during the commissioning stage if the two halves are not fully bedded out.

Flash often forms when moulding very low viscosity materials, as the melt can flow in to the thinnest of gaps.

Example of Flash

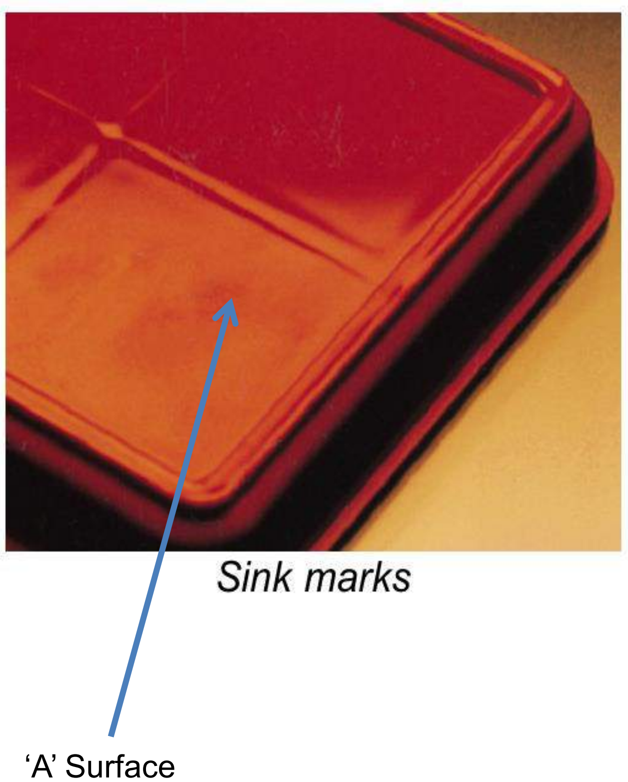
Solutions

1. Ensure the mould is bedded out properly.
2. Reduce the injection speed.
3. Reduce the maximum cavity pressure during injection or hold.
4. Reduce the melt temperature.
5. Check rigidity of the mould and machine platen.
6. Check material viscosity is within specification.
7. Consider running the mould on a machine with higher clamp force.
8. Check sufficient clamping force has been set.
9. Check that there is no debris between tool faces holding the tool off.

Trouble Shooting Guide for Injection Moulding

Sink Marks

Sink Marks occur on the ''A'' surface of the moulding or bubbles through slightly thicker regions of the part.



Solutions

1. Check if melt cushion is less than 5 mm and increase if necessary.
2. Check function of the non return valve.
3. Experiment with increasing hold pressure time.
4. Increase the hold pressure.
5. Reduce the temperature of the mould.
6. Reduce the melt temperature.
7. Reduce the injection speed.
8. Extend the cooling time.
9. Increase the gate size.
10. Check wall thickness in the gate area.
11. Reduce the thickness of surface details such as ribs and bosses.

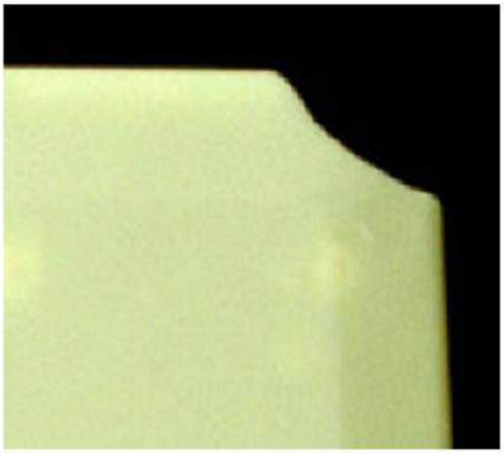
Trouble Shooting Guide for Injection Moulding

Short Mouldings

The plastic moulding has sections missing, usually in regions remote from the injection point or in thin-walled regions, because the mould was not completely filled.

Common causes are the metered shot size is too small, the non-return valve is leaking or the injection pressure is set too low.

If the machine is running on maximum fill pressure, then the pressure drop through the feed system and cavity could be



Short moulding

Solutions

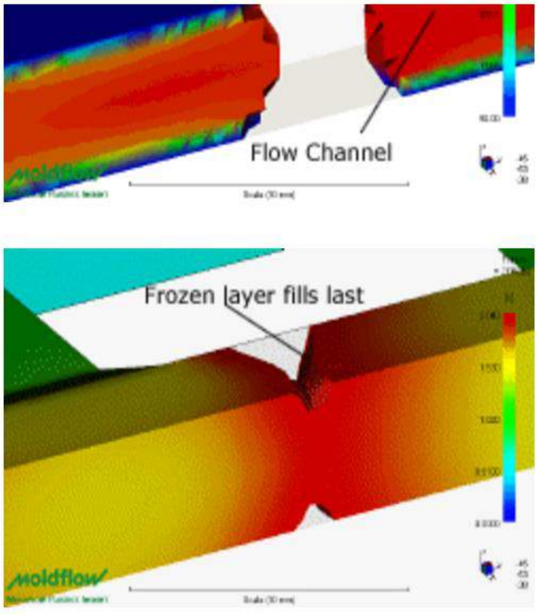
1. Increase the shot volume.
2. Switch over later from injection pressure to hold pressure.
3. Increase the injection speed.
4. Increase the temperature of the melt and mould.
5. Improve venting at the end of the flow path.
6. Choose an easier flowing material.
7. Increase the gate size.
8. Check the actual wall thickness of the cavity.
9. Move the position of the gate.
10. Incorporate flow leaders .
11. Check for material leaking from nozzle.
12. Reduce clamping force to allow gas to vent out.

Trouble Shooting Guide for Injection Moulding

Weld or knit Lines

Visible lines formed on the surface of the moulding, usually located between gate points or downstream of apertures or other obstacles.

Weld lines are formed when separate melt fronts travelling in different directions meet and can be visible and/or reduce the part strength.



Weld line simulation

Solutions

1. Increase the injection speed.
2. Increase the hold pressure.
3. Increase the melt temperature.
4. Optimise hold pressure switch over point.
5. Ensure adequate venting in the region of the weld line.
6. Increase the mould temperature.
7. Investigate moving the gate point.
8. Change master-batch check compatibility.

Trouble Shooting Guide for Injection Moulding

Parts stuck in the mould

Parts tend to stick in one half of the mould.

Parts tend to stick on one half of the mould.

Part stuck on the moving half.

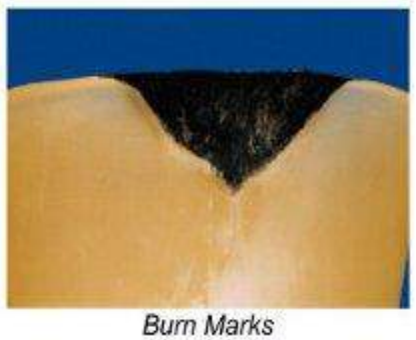
Solutions

1. Decrease injection pressure.
2. Decrease the holding pressure.
3. Minimize cushion.
4. Check for undercuts or insufficient draft particularly on textured surfaces.
5. Stuck on the Fixed Half:
6. Increase close time.
7. Decrease Moving half temperature.
8. Stuck on the Moving Half:
9. Decrease close time.
10. Decrease Fixed half temperature.
11. Polish Surface of the Tool.
12. Vapour Blast the surface of the Tool.

Trouble Shooting Guide for Injection Moulding

Burn Marks Gas Traps

Burn marks. Air and gasses in front of the flow front can not escape due to insufficient venting and are compressed to the point of ignition.


Burn Marks

Variable wall thickness can cause the melt to encirculate air and gasses at the flow front resulting in gas trap.

Solutions

1. Check tool faces are clean
2. Reduced injection speed.
3. Reduce injection pressure.
4. Improve venting in the area of the burn mark.
5. Decrease the melt temperature in case of degradation.
6. Check thermocouples and check band heaters.
7. Reduce clamping force if possible

Gas Traps

Gas Traps or Air Traps occur when the melt races around a local area, the resultant gas trap can range from incomplete filling to burning due to the Diesel effect.

Solutions

1. Change the flow pattern by means of flow leaders or alternative gate points.
2. Move the position of the gate.
3. Adjust the wall thickness profile.
4. See also solutions for burn marks.

Trouble Shooting Guide for Injection Moulding

Black Specs

Black/Brown specs or deposits are visible on the surface of the moulding, normally caused by contamination or degradation of the material.

Black Specks

Solutions

1. Reduce the melt temperature of the Polymer.
2. Reduce the screw speed and/or back pressure
3. Clean the moulding machine thoroughly , specifically the hoppers hopper throat.
4. Check for impurities , leaks or dirt deposits in the feed stock hopper supply.
5. Check the shot weight. The entire shot weight should not be less than 15-25% of the maximum capacity of the moulding machine.
6. Check the compatibility of pigment or master-batch with the base polymer.
7. Reduce the time in which the material is openly exposed to the ambient conditions.

Trouble Shooting Guide for Injection Moulding

Ejector Marks

After ejection from the mould, ejector pins have clearly left visible marks on the surface of the moulding in the form of depressions, differences in gloss or stress whitening.

Ejection Pin Marks

Solutions:

1. Ensure ejectors finish flush with the surface of the mould.
2. Optimize the changeover to hold pressure.
3. If sink marks are visible on the ejector points, increase the hold pressure.
4. If differences in gloss are visible on the ejector marks reduce the hold pressure.
5. Reduce the injection speed to minimise injection pressure.
6. Extend the cooling time and/or changeover time.
7. Ensure accurate location and movement of ejector pins.
8. Reduce the hold pressure.
9. Reduce the temperature of the mould core.
10. Use a mould release agent.
11. Check the mould for undercuts and draft.
12. Ensure adequate venting of the mould.
13. Reduce the melt processing temperature.
14. Install additional ejectors and/or enlarge existing ones.
15. Polish tool to ease ejection.

Trouble Shooting Guide for Injection Moulding

Nozzle Drool

Nozzle Drool String might remain from previous shot transferred into the new shot and usually evident on the surface of the moulding.

Solutions

1. Reduce nozzle temperature.
2. Reduce material temperature.
3. Minimize cushion.
4. Delay sprue break time.
5. Decrease mould open time.
6. Use nozzle with smaller diameter orifice.
7. Use decompression to hold the melt back from the nozzle.
8. Ensure that the material has been correctly dried.
9. Consider Fitting Shut off Nozzle or Tip.

Trouble Shooting Guide for Injection Moulding

Cold Slugs

A small amount of melt solidifies at the gate point before the start of injection and is pushed into the mould cavity via the melt stream. Evidence of a cold slug normally occurs close to the injection point.

Solutions

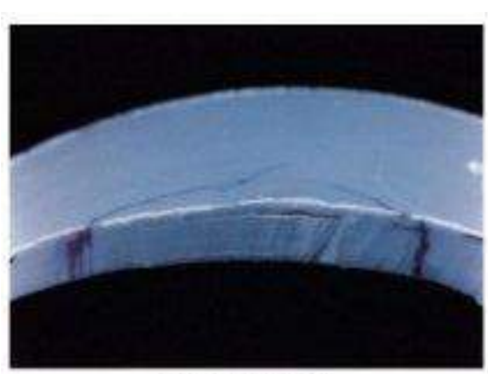
1. Increase the temperature of the nozzle slightly.
2. Reduce the back pressure.
3. Increase the retraction of the screw - decompression.
4. Add a “cold slug well” to the cold runner feed system.
5. Consider increasing the nozzle / gate diameter.
6. Consider use of a shut-off nozzle.
7. Reduce the time that the nozzle is in contact with the cold mould.
8. Utilise Cold Slug Removal Feature on the machine.

Trouble Shooting Guide for Injection Moulding

Delamination/Splay

Delamination refers to the splitting open/peeling off of the surface or separation of the compound components and is usually due to excessive shearing of the melt breaking down the polymer structure.

Splay is similar but the delamination appears like radial cracks or lines on the moulding surface.



Delamination

Solutions

1. Reduce the injection speed and increase the processing temperatures.
2. Check the settings data against the last successful production run.
3. Clean the machine thoroughly when there is a change of plastic material.
4. Check compatibility of the current master-batch or other additives.
5. Consider increasing the gate size to reduce shear rate.
6. Consider adding a hemispherical gate well opposite the gate to reduce shear and stress in the immediate area.

Trouble Shooting Guide for Injection Moulding

Discolouration

Differences in colour or colour streaks in the surface of the moulding.

The cause is usually due to a non-uniform distributions of the pigments during plasticisation but can also be due to non-compatibility of the polymer and master batch.

Solutions

1. Check compatibility of the master batch with the base polymer.
2. Check the screw and barrel for defects.
3. Increase the speed of injection.
4. Reduce and increase the screw speed. Evaluate the effects on streak formation.
5. Increase back pressure during plasticisation to improve mixing.
6. Confirm the temperature of the melt is within specification.
7. Confirm the temperature is within limits of the master batch.
8. Check the hot runner temperature control and operation of thermocouples.
9. Check whether the shot weight, barrel volume and residence time is within guidelines.

Trouble Shooting Guide for Injection Moulding

Inconsistent Melts

Inconsistent melting and pigmentation.

A homogeneous melt is crucial for maintaining part quality.

An inconsistent melt may be evident by poor melting of the granules or poor colour dispersion.

Solutions:

1. Barrel temperatures too low, particularly at rear zone.
2. Increase screw speed.
3. Increase back pressure.
4. Check for proper screw compression ratio.
5. Check for wear of the screw or barrel.

Trouble Shooting Guide for Injection Moulding

Gloss Differences

The surface of the plastic moulding exhibits different levels of gloss even though the mould surface finish is uniform.

A homogenous melt is crucial for maintaining part quality. An inconsistent melt may be evident by poor melting of the granules or poor colour dispersion.

Solutions

1. Experiment with increasing the mould temperature.
2. Increase the melt temperature.
3. Increase the hold pressure.
4. Check and optimise hold time.
5. Optimise the point at which the changeover to hold pressure is made.
6. Optimise the injection rate.
7. Improve consistency of the melt by increasing the back-pressure and / or screw speed.

Trouble Shooting Guide for Injection Moulding

Jetting

From the gate point, the surface of the plastic moulding exhibits a snaking, often through rough or matt strand of polymer melt?

Jetting often occurs when the melt is directed from the gate into the open cavity at high local velocity.

Ideal conditions for jetting usually occur when the melt flows through a small gate into a thick section or is fired between the walls of the injection mould.

Solutions

1. Reduce the speed of injection.
2. Incorporate an obstacle in front of the gate like a pin.
3. Enlarge the gate to reduce shear.
4. Move the gate to an adjacent area where the melt is forced to flow around an obstacle or against a wall
5. 

Example of Jetting

Trouble Shooting Guide for Injection Moulding

Moisture/Silver Streaks

Silver/Moisture streaks are distinct streaks, usually radiating from the gate and are often caused by excess moisture, air introduced or degradation during plasticisation of the melt in the barrel of the machine.

Many plastics absorb moisture from the atmosphere; how much they absorb depends on the type of resin. Moisture in the granules, even if it is only surface condensation, can cause problems in parts including, poor surface finish, processing issues, increased flash or loss of mechanical properties such as impact and tensile strength in some materials.

Solutions

1. Increase back pressure.
2. Decrease melt temperature.
3. Check the mould cooling circuit for leaks onto the mould face.
4. Check the packaging of the plastic granules for damage.
5. Check the moisture content of the granules.
6. Ensure pre-drying at recommended temperature and time.
7. Reduce the quantity of granules held in the feed hopper.
8. Check raw material storage conditions.

Most engineering materials require between 2 and 4 hours drying to reach typical acceptable moisture content for moulding: 0.02 %.
Hot air dryers are not always Effective specially in a humid environment; therefore dehumidified-air drier systems are the preferred method.

Trouble Shooting Guide for Injection Moulding

Tiger Lines

Tiger lines are radial shadows on the surface of the moulding.

Caused by pulsating polymer melt flow.

Some compounds naturally exhibit a slip stick flow effect that can show up as tiger stripes, but a common cause is worn screw, barrel or check ring.

Solutions

1. Increase the temperature of the melt.
2. Increase the temperature of the mould.
3. Check the temperature profile and operation of the hot runner system.
4. Enlarge the gate cross section.
5. Choose an easier flowing material.
6. Check for worn screw, barrel or check ring.

APPENDIX 1

MATERIAL SELECTION GUIDE

1. GET IN THE RIGHT GROUP
WHAT IS MOST IMPORTANT TO THE APPLICATION?

GENERAL GROUP CHARACTERISTICS

AMORPHOUS THERMOPLASTICS

- Soften over a wide temperature range
- Good formability
- Transparency
- Poor chemical resistance
- Bond well using adhesives or solvents
- Prone to stress cracking
- Poor fatigue resistance
- Structural applications only (not suitable for bearing and wear)

POTENTIAL MATERIAL CHOICES

- ABS
- Acrylic
- Kydex®
- Noryl®
- PETG
- Polycarbonate
- Polystyrene (HIPS)
- Polysulfone
- PVC
- Radel R®
- Ultem®

SEMICRYSTALLINE THERMOPLASTICS

- Sharp melting point
- Poor formability
- Opaque
- Good chemical resistance
- Difficult to bond using adhesives or solvents
- Resistant to stress cracking
- Good fatigue resistance
- Good for bearing and wear (as well as structural applications)

POTENTIAL MATERIAL CHOICES

- Acetal
- HDPE
- LDPE
- Nylon
- PBT
- PEEK
- PET
- Polypropylene
- PPS
- PTFE
- PVDF (Kynar®)
- UHMW-PE

IMIDIZED MATERIALS

- Best physical properties above 400°F
- Best temperature resistance
- Best bearing and wear capabilities
- Good chemical resistance

POTENTIAL MATERIAL CHOICES

- PAI (polyamide-imide)
- Vespel® Polyimide Shapes

This selector guide is intended to help you review the needs of your particular application and determine a few material candidates that can then be tested.

Your trusted partner for
Plastic Injection Moulding