6 Defect Identification and Troubleshooting

6.1 Defects Table

The following table contains a list of some of the most frequent molding problems, their possible causes and methods of rectify. Although the description is made to the best of our knowledge, we cannot guarantee that the problems will be successfully rectified using these measures. The list is to be used as a general guideline. The operator should make a detailed analysis of the defect on-site.

Table 6: List of defects

Sign	Possible cause	Resolution
Sink marks on the molded part	Cushion too small. Sink marks close to the gate or in thick-wall areas.	Shot-size increase Increase backpressure, optimize backpressure time. Increase backpressure. Decrease mold wall temperature. Increase melt temperature. Decrease shot speed. Adjust gate diameter.
	Sink marks far from the gate or in thin-wall areas.	Optimize backpressure time. Increase backpressure. Increase shot speed. Increase melt temperature. Increase mold wall temperature. Adjust gate diameter.
	Deformation during ejection.	Increase cooling time.
	Melt too cold, resulting in too high shear stress.	Increase melt temperature. Increase hot runner temperature.
	Melt too hot, resulting in damage to the molded part due to high temperature.	Decrease material setpoint temperature. Decrease hot runner temperature (do not decrease nozzle temperature too much, could cause freezing-off). Decrease shot speed. Adjust gate diameter.
	Temperature sensor of the hot runner (inlet bushing, manifold, nozzle).	Temperature sensor position: Sensor too far from heating, in a cold area. Place the sensor closer to heating. Temperature sensor function: Correct sensor type (K, J)? Correctly calibrated controller (K, J)?

Defect Identification and Troubleshooting

Sign	Possible cause	Resolution
Burnt smudges (brown or silver)	Cushion too large.	Reduce cushion.
	Shear stress on the actuator too high.	Decrease screw rotation speed.
	Long residence time in plastification unit under critical conditions.	Decrease cycle time. Increase delay at feeding. Use a smaller plastification unit.
	Shear stress at gate too high.	Decrease shot speed. Check / adjust gate diameter. Remove transitions with sharp edges from the hot runner system.
	Material too dry.	Check/reduce temperature/drying time. See drying instructions from material producer.
	Differences in hot runner diameter too big Channel diameter for the melt too big (long residence time).	Clean the hot runner. Adjust the channels for the melt in the hot runner. Wrong, excessive dimensioning.
	Contact between hot runner nozzles and manifold, resulting in overheating (noticeable on the high load of the heating zone).	Check the cut out and the contact surfaces of the hot runner and the manifold vs. the injection mold. Increase the isolation gap (Joint Z). Cut out dimensions as per the customer drawing
The cavity stops filling with material after more than 5 shots	Check the cut out and the contact surfaces or the hot runner and the manifold vs. the injection mold. Cold nozzles, frozen gate.	Check if the nozzle does not bear on the cut out (take into account thermal expansion), correct, if needed. Check injection mold, machine nozzle and inlet bushing for tightness. Check the function of heaters and replace them, if needed. Check the temperature at mold gate using a pyrometer. Increase temperature at mold gate by 28-56°C (50-I00°F) above operating temperature until all cavities are filled, and decrease to normal operating temperature. Caution is required: not all plastics withstand this temperature increase. If this is the case do not set such a high temperature.

Defect Identification and Troubleshooting

Sign	Possible cause	Resolution
Dark spots	Granulate contaminated.	Prevent contamination. Clean the plastification unit and the hot runner.
	Melt temperature too high.	Decrease melt temperature. Decrease hot runner temperature. Decrease screw rotation speed. Decrease hot runner temperature (avoid nozzle freezing). Decrease dynamic pressure.
	Too long melt residence time in the system.	Decrease cycle time. Increase delay at feeding. Use a smaller plastification unit. Check the channel diameter in the hot runner.
	PC processing.	When production is interrupted, decrease hot runner and actuator temperature to max.160°C (320°F). If temperature is to drop in general, it is essential to flush the hot runner and the plastification unit in advance (natural PP or natural PE).
	Wearing, dead spots.	Check plastification unit, gate system and hot runner for contamination, wearing and dead spots.
Stringing in the gate area	Melt temperature too high.	Decrease melt temperature. Decrease hot runner temperature. Decrease screw rotation speed. Decrease hot runner temperature (avoid nozzle freezing).
	Gate area too hot.	Decrease mold temperature. Check contact tightness between nozzle tip and mold. Check mating between nozzle tip and mold. Check isolation cap thickness. Use colder nozzle / cone point insert. Temperature sensor position: Sensor too far from heating, in a cold area. Place the sensor closer to heating. Temperature sensor function: Correct sensor type (K, J)? Correctly calibrated controller (K, J)?
Part deformation	Injection pressure too low.	Increase injection pressure. Increase cooling time. Increase injection mold surface temperature. Decrease regrind percentage. Increase gate diameter.
	Afterpressure time too short.	
	Heating defect.	
Smudges	Material too cold.	Increase material temperature. Increase injection mold surface temperature. Increase gate or runner channel.
	Gate or channel too small.	
	Injection mold too cold.	
	Nozzle hole too small.	
Matt surfaces	Material / injection mold too cold.	Increase material / injection mold temperature.
Molded parts are brittle	Material / injection mold too cold / hot. Injection speed too high. Gate or flow channel too small.	Increase material/ injection mold temperature. Decrease injection speed. Increase/decrease injection mold temperature. Increase gate or flow channel.

Defect Identification and Troubleshooting

Sign	Possible cause	Resolution
Actual temperature does not reach the required value:	Heating power does not cover heat losses due to support elements, center rings and inlet bushing.	Verify correct Hot Runner System installation in mold.
	Nozzle heating in contact with the injection mold.	
Unstable value of actual temperature	Unstable value of actual temperature.	Check temperature controller Notify Services Verify correct assembly dimensions.
	Unsuitable sensor position.	
	Defective sensor.	
	Unstable system contact, synchronization with injection cycle.	
Max. heating power cannot be reached	Weak temperature control. Heating power limited by manual setting.	Increase heating power set manually.
Gate freezing	Gate too small.	Check for presence of cold / non-plastified material. Check injection mold and processing temperature. Check for presence of foreign material. Check temperature controller. Notify Services.
	Contact surfaces too large.	
	Foreign body in the gate.	
	Nozzle run, cold plug.	
	Defective sensor or heating.	
	Heating in contact with the mold.	
	Irregular injection cycle.	Mold temp too low.
Plastic leakage	Temperature too high.	Check temperature. Increase screw decompression. Check pressure.
	Gate too large.	
	Matt places around the gate or at sharp edges.	
	Contact surfaces too small.	Replace tips
Matt places around the gate or at sharp edges	Shot speed too high.	Graduated injection speed profile slow – medium – fast.
	Melt temperature too low.	Increase injection pressure. Increase mold temperature.
	Mold temperature too low / uneven.	Increase mold temperature.
	Melt face speed too high.	Significantly decrease shot speed before the flow face reaches sharp edges in the mold.