Injection Molding Sink Marks

Common Cause and Practical Solution

Injection molding sink marks are a common defect in plastic injection molded parts. They usually appear as small depressions on the surface, especially behind ribs, screw bosses or thick sections. One common cause is excessive local wall thickness, which slows cooling and increases internal shrinkage. This article explains why sink marks occur and how optimizing the part structure can help reduce them.

What Causes Sink Marks in Injection Molding?

One common cause of sink marks is excessive local wall thickness.

When a plastic part has a thick rib, solid boss or other concentrated area, the thicker section contains more material and takes longer to cool.

During cooling, the plastic continues to shrink. If the outer surface has already solidified while the inside is still shrinking, the surface may be pulled inward and form a visible sink mark.

In simple terms:

Thicker section → slower cooling → greater internal shrinkage → sink mark

Why Do Ribs and Bosses Often Cause Sink Marks?

Ribs and bosses are widely used to improve the strength and assembly performance of plastic parts. However, if they are too thick, they can create excessive material accumulation.

For example, a plastic housing may have a normal wall thickness of 2.5 mm, while a thick rib or solid boss creates a much thicker local section. This difference in thickness can cause uneven cooling and shrinkage.

Therefore, when a sink mark appears behind a rib or boss, it is important to check the internal structure and local wall thickness, rather than simply increasing injection pressure.

A Common Solution: Optimize the Part Structure

If excessive local wall thickness is confirmed as the main cause, a common solution is to reduce material accumulation. Possible improvements include:

- • Reducing excessive rib thickness
- • Using cored-out bosses instead of solid bosses
- • Avoiding unnecessary thick sections
- • Maintaining more uniform wall thickness
- • Improving the transition between thick and thin areas

The goal is to make the part cool more uniformly and reduce differences in material shrinkage.

For new products, optimizing the structure during the design stage is usually more effective than trying to solve the problem later through molding parameters.



Sink marks behind ribs and bosses, and the structural optimization approach.

What If the Mold Has Already Been Made?

When the mold is already completed and structural changes are difficult, molding conditions can also be evaluated. Important factors include:

- • Holding pressure
- • Holding time
- • Gate size and location
- • Gate freeze time
- • Mold temperature
- • Cooling efficiency

Increasing holding pressure or holding time may reduce sink marks when insufficient packing is part of the problem. However, excessive pressure can cause other defects such as flash, internal stress or dimensional changes.

Therefore, the goal is to find a stable processing window, rather than simply increasing pressure.

Sink Marks in PA6 and PA66 Injection Molding

For engineering plastics such as PA6 and PA66, material characteristics also need to be considered. Factors such as glass fiber content, material shrinkage, mold temperature, processing temperature and part geometry can all affect dimensional stability and surface quality.

For applications requiring higher strength and dimensional stability, the appropriate modified nylon grade should be selected according to the actual product requirements.

Conclusion

Injection molding sink marks should not always be solved by increasing injection pressure.

When a sink mark appears behind a rib, boss or thick section, the first step is to check whether excessive local wall thickness is causing uneven cooling and internal shrinkage.

A practical improvement approach is:

Optimize part structure → reduce material accumulation → improve cooling → reduce shrinkage differences

For existing molds, holding pressure, holding time, gate design, cooling conditions and material selection can be evaluated together.

At Dongguan JiCheng techinherit Co., Ltd., we provide modified nylon materials and injection molding support to help customers address common plastic molding issues such as sink marks, warpage, short shots, floating fibers and dimensional instability.