

# What are the eight conditions of the injection moulding process?

The eight conditions of the injection moulding process are injection rate, injection volume, screw injection position, injection time, cooling time, screw speed, anti-delay volume, and residual material volume. The following classification introduces the performance of these eight processes.

## Introduction:

The injection moulding process is a complex manufacturing method used to produce parts by injecting molten material into a mould. It's widely used for manufacturing a variety of parts, from the smallest components to entire body panels of cars. The quality, efficiency, and consistency of the injection moulded parts are significantly influenced by several key process conditions. Here are the eight fundamental conditions of the injection moulding process:

## Injection rate:

Injection rate refers to the screw forward to fill the plastic melt into the mould cavity when the rate, generally with the unit between the flow of the injection mass (g / s) or the screw forward speed (m / s) that it and the injection pressure are one of the important conditions of the injection conditions. Different injection speeds can lead to different results.

In low-speed injection, the material flow rate is slow, the melt from the gate begins to gradually flow to the far end of the cavity, before the material flows into a spherical, first into the cavity of the melt first cooled and slowed down the flow rate, Near the cavity than the wall of the part of the cooling of the wall into a high elasticity of thin shell, and away from the cavity wall of the part of the viscous state of heat flow, and continue to extend the end of the flow of spherical, to fill the cavity, the thickness of the cooled shell to increase the hardening. This slow-speed mould melts into the cavity for a long time, cooling makes the viscosity increase, and the flow resistance also increases, so it must use a higher injection pressure mould.



## **Injection volume:**

The injection volume for the products and the main flow channel manifold and so on, when added together the total mass, if its value is less than the [moulding plastic machine](#)'s larger injection volume, is theoretically can be moulded. However, in general, the injection volume should be less than 85% of the rated injection volume of the plastic injection molding machine. However, if the actual injection volume used is too small, the plastic will be thermally degraded due to the long residence time in the cylinder. To avoid this phenomenon, the actual injection volume should be more than 30% of the rated injection volume of the [injection moulding machine](#), therefore the general injection volume should be set between 30% and 85% of the rated injection volume of the plastic molding machine.

## **The injection position of the screw:**

the injection position is one of the most important parameters in the [injection moulding process](#), the injection position is generally based on the total weight of the plastic part and the condensate is the outlet material to determine, and sometimes needs to be used according to the type of plastic, mould structure, product quality and so on to reasonably set up the position of injection of the backlog section.

## **Injection time:**

The injection time is the time to apply pressure to the screw, including the flow of plastic, mould filling, and holding pressure time required, so the injection time, injection speed, and injection pressure are more important injection moulding conditions, how to find the correct injection time can be carried out in two ways: appearance setting method and weight setting method.

Although the injection time is very short and has little effect on the moulding cycle, the adjustment of the injection time has a significant effect on the pressure control of the gates, runners, and cavities. An appropriate injection time can help the melt to achieve ideal filling and

is also very important in improving the surface quality of the product and reducing dimensional tolerances.

## **Cooling time:**

The cooling process is started by the injection rather than after the injection is completed, and the length of the cooling time is based on ensuring that the plastic part is shaped to be able to open the mould and take out the settings. In general, the cooling time accounts for 70-80% of the cycle time.

## **Screw speed:**

Screw speed affects the path and shear effect of the resin transported and plasticized in the screw, which is an important parameter affecting plasticizing capacity, plasticizing quality, and moulding cycle time. As the screw speed increases, the plasticizing capacity increases, the uniformity of the melt temperature increases and the plasticizing effect decreases. Screw speeds are generally 50-120 rpm.

For heat-sensitive plastics such as PVC, POM, etc., a lower screw speed can be used to prevent material degradation, and for plastics with high melt viscosity, a lower screw speed can also be used.

## **The amount of anti-extension:**

that is, the amount of screw loosening, screw metering in place, and then a straight line back a distance, so that the space of the melt in the metering chamber increases, the internal pressure drops to prevent the melt from the metering chamber to the outflow, this backward action is called anti-flow delay, backward distance is called the amount of anti-extension or anti-flow delay stroke.

Anti-flow delay also has another purpose, which is to reduce the pressure in the nozzle runner system when the nozzle is not returned for pre-forming, reducing internal stress and making it easier to remove the main runner when the mould is opened. The setting of anti-extension depends on the viscosity of the plastic and products, generally 2-3mm, too large anti-extension will make the melt in the metering chamber entrained bubbles, seriously affecting the quality of the product, and the viscosity of the material can not be set anti-extension.



## Residual Material:

After the end of the screw injection, and do not want to shoot all the melt from the head of the screw, but also want to keep some, the formation of a residual material. In this way, on the one hand, to prevent mechanical collision of the screw head and nozzle contact; on the other hand, the amount of residual material to controls the injection volume of the repeatability of the accuracy to achieve the purpose of stabilizing the quality of [injection moulded products](#) (residual material is too small, it can not achieve the purpose of cushioning; too large a large amount of residual material accumulated too much). The general amount of residue is 5-10mm.

## Final Thought

These conditions are interdependent, and adjustments in one can affect the others and the overall quality of the finished product. Mastery over these conditions enables manufacturers to produce high-quality parts efficiently and consistently.