Injection Mold Cooling Designs

The design of the injection mold cooling system is very important. The cooling time takes up 70% to 80% of injection molding cycle. A well-designed cooling system can shorten the molding time and significantly improve the productivity. Poor design of a cooling system will extend molding time and increase production cost. In addition, the injection mold temperature has a great influence on the mold shrinkage, dimensional stability, deformation, internal stress and surface quality.

So what are the factors that matter to the cooling effective?

**Plastic wall-thickness**
Parts with thicker walls would need longer cooling time. Generally, the cooling time is approximately proportional to the square and the thickness of plastic parts. If possible, propose to the part designer to minimize the wall thickness.

**Mold steel**
The higher the thermal conductivity of the injection mold steel, the better for heat transferring and reduction in cooling time. In practice, the injection mold shop usually utilizes copper to replace steel whereas on the cooling line, this is not possible to do.

**Cooling line layout**
The closer the mold cavity goes to the cooling pipes, the greater the diameter of cooling pipes, the better the cooling effect, the shorter the cooling time is going to be. The designer should consider all of these elements to achieve maximum cooling effect.

**Coolant**
The nature of the coolant could be different – frequently-used coolants are oil and water. Viscosity and thermal conductivity of the coolant also affects the heat conduction effect of the plastic injection mold. The lower the cooling fluid viscosity, the higher the thermal conductivity - the lower the temperature, the better cooling effect.

**Cooling system design rules:**
* Insure cooling efficiency, achieve the shortest the cooling line all the while producing quality parts.
* Insure uniform cooling to avoid deformation.
* Ease of manufacturing
Some examples of injection mold cooling designs
If possible, maximize the number of cooling channels and design the diameter of the cooling channel to be as large as possible. Diameter of cooling channels usually are 6-12mm.

Cooling channels layout must be reasonable. When the wall thickness of part is uniform, the distance of each channel to the surface of parts should be even, which means the layout of channels should follow the actual geometry of the part. When the thickness of the part is not uniform, then thicker wall areas need more cooling - the injection mold cooling channel should then be closer to the part to enhance the cooling effect.

Usually, temperature of the sprue gate area are highest, so starting the cooling process there would achieve the best cooling effect.

Re-think cooling, that is heat removal, as "Thermal Management" and not simply "Cooling". The engineering of the Thermal System is not trivial but is often over-looked. This phase of the process cycle is the longest accounting for 70-80% or more of the time. This means there is a large financial incentive to reduce this and increase profit. However, the traditional way of treating cooling as less critical discounts that dimensional stability and quality is driven during this phase. Removing parts early leads to many issues so the balance is finding the best time and the best method to optimize cooling.

Look at cooling as "The goal of cooling is to remove heat from the part in an advantageous way to result in a low stressed part with the best dimensional and physical properties." Often, this is treated as an even rate while in truth, the cooling may need to be differentiated to pull the heat from difficult areas of the part and allow for a steady state result. This requires a thorough engineering approach including thermocouples to measure the thermal performance of the injection mold, using water circuits and manifolds to control temperatures, and, perhaps, tool materials with tailored thermal properties. This list is not inclusive of a fully-engineered thermal system but is a thought starter. Using this approach along with good engineering practices will go a long way in moving from designing injection molds (rule of thumb) and engineering tools (using solid practices) to getting efficient tools, improving processing margins, and generating a better profit margin.

TPG makes every effort to insure that the information contained herein is accurate - however, we accept no liability for the content of this piece, or for the consequences of any actions taken on the basis of the information provided.