How to Design an Injection Molding Tool to Reduce Cycle Time:

Injection Molding Tooling

Aside from designing an injection molded part, tool design is equally important. Having a properly designed tool is important because one; a poorly designed tool undermines the quality of the part and two; the tooling process is important to the success of the part itself. In this article we will examine what a tool is, some best practices for tool making and your options as far as designing a tool goes. If you are not ready for tool design yet and still need to design a part, follow the link here: {Insert link}

TOC:

i) Venting

ii) Gating

iii) Part line

iv) Material

What is an injection molding tool?

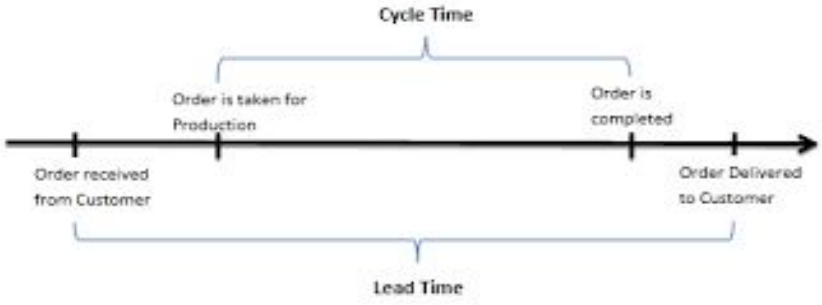
A “tool” is also called a *mold* or a *die*.

The process is as follows: Raw material (aka plastic resin) is top-loaded into what is called a hopper. The hopper feeds the injection molding machine via gravity into a heated barrel equipped with a reciprocating screw that melts the plastic, pushing it and injecting the melted plastic into what you see to the left. This is the mold. The plastics forms and cools in the contour of the mold which is then ejected to form the finished part.

As you can see, having the right mold design is absolutely crucial because this is where the entire process is happening. A faulty design in the tool will result in having to repeat production methods which are both time-consuming and costly.

Best practices for tool designing:

Just like there are best practices for *part design*, there are best practices for tool design. These best practices are implemented to reduce the risk of failure in the manufacturing process. Some of the common issues associated with poor tooling design are poor material flow, inefficient or irregular cooling times, burnt parts, damaged machines or tools, excessive flash; the list goes on. What does that mean for you? This means that the part will not come out as planned, or a whole order could be unviable. Costs will begin to accrue as *cycle time** increases.



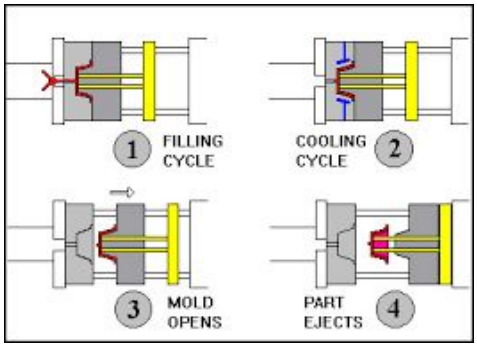
Cycle Time: Total time to complete production of one unit (work process based)

Reducing the cycle time will help cut costs in the long run and will add to your bottom line as a business. To reduce cycle time, pay extra attention to the way your part and tool are designed and make sure the tool is regularly maintained properly! The following design elements will help deter complications in the injection molding process and keep cycle time at an efficient pace.

Design elements of a mold:

Venting:

Venting is regarded as one of the most important elements of mold design.

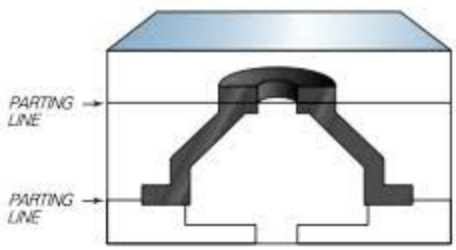


Venting is where air and gas escape from the mold cavity as it's filled with hot liquid plastic. This process also allows the material to completely fill the cavity, as opposed to air or other gases that can potentially fill the cavity resulting in a short shot or flash.

The biggest challenge with venting is allowing enough room for air and gas to escape without compromising the final product. Failing to vent properly could result in several issues, including:

- Gas burns
- Plateouts
- Flash
- Short shots
- Oil buildup

Part line location:



The parting line is the location on the part where the two halves of the mold meet for injection and split for ejection. This is where most flash is likely to occur from multiple factors like force from injection or an improper fit of the two halves. Flash at the parting like can be avoided by close inspection of the tool as well as investing in a high quality tool.

Material Selection:

For most jobs you will have two options for the material your mold is made of: aluminum and steel. Aluminum is often thought of as a low-cost option and has been said to have limitations as to its applications. Although aluminum is a weaker metal, there are some advantages to choosing it as your mold base and core:

1. Aluminum has a 5x better heat transfer rate. With a better conductivity, the position of cooling lines becomes less critical thus allowing for more mold components to be placed.

2. Aluminum has demonstrated a reduction in production cycle time by up to 30%.
3. The heat treatment process for aluminum is much better than that of steel since it does not have to be sent out to be annealed. Annealing is a heat treatment process for metals to remove internal stresses and to toughen the material.
4. Aluminum is more cost effective than steel.

Although aluminum molds have its advantages, it is only capable of producing between one to two thousand units, whereas steel can guarantee a minimum of 50,000 units. So if you have a small production run, aluminum will certainly be your preferred tooling method at a fraction of the cost of steel. (via Jaycon Systems:Medium)

P20 steel is the most commonly used material for molds in the plastics manufacturing industry due to its unique characteristics. Some of its main properties include:

1. Hardness;
2. Good polishing ability;

3. Heat treatment possibility after machining;
4. Good corrosion resistance;
5. Good wear and tear resistance;
6. Good thermal stability.

P20 material is a versatile low alloy steel that is characterized by its hardness and moderate strength level. It is typically sold in pre-hardened condition at a hardness of approximately 40 to 50 HRC. Having a homogeneous mixture of alloys allows the P20 steel to obtain uniform hardness within the whole material. In P20 steel, the presence of metals such as chromium and nickel enhances its strength and hardness, making it more desirable than other materials. (via Jaycon Systems:Medium)

That concludes our guide to reducing cycle time with effective tooling design. Want more help on designing a tool or related injection molding services? Visit us at www.plasticsplus.com and get started on your project!

(sources:Revparts;Jacob Schwartz. Jaycon Systems)