EIGEN

Your Partner for Precision

7 MORE DESIGN GUIDELINES FOR INJECTION MOLDING

Injection molding is one of the most widely used process for manufacturing numerous products. Manufacturing of small components such as a plastic covers to large components such as body panels for trucks are taken care of in the injection molding process. Once the designing of a product takes place, the appropriate mold is made and the appropriate machining is used to mold the needed products.

▶ Inception to completion of injection molding

The designing team gets the details about the parts or products, materials to be used and the quantity as well. After the completion of a design, the precise tool and mold is made for the same. For example, if it is a thermoplastic or thermoset plastic, then it is fed into the pre-heated barrel and after mixing it is led to the mold cavity where it is kept to cool down. Once the product hardens, it is removed and then final touches are given to make it suitable for client's requirements.

There are thousands of companies that are looking to get top quality results with the injection molding, and a lot has been written in this regard. However, to shed more light on this area, here are 7 more design guidelines for injection molding.

1. Thickness of the wall

Injection molding is all about precision, and so the manufacturers need to make sure that there is uniform wall thickness achieved throughout the product. Some of the many benefits that this provides is minimal warp, sinking, residual stressing, and this also improves cycle times and mold fills. There are various materials that can be used in the injection molding process depending upon the client requirements, and every material will have a recommend wall thickness to be followed for excellent results.

The thick areas need to be redesigned so that a more uniform wall design can be achieved. In order to get a uniform thickness in wall, it is important to select the right material and follow thickness guidelines. Some of the standard material thickness recommend are:

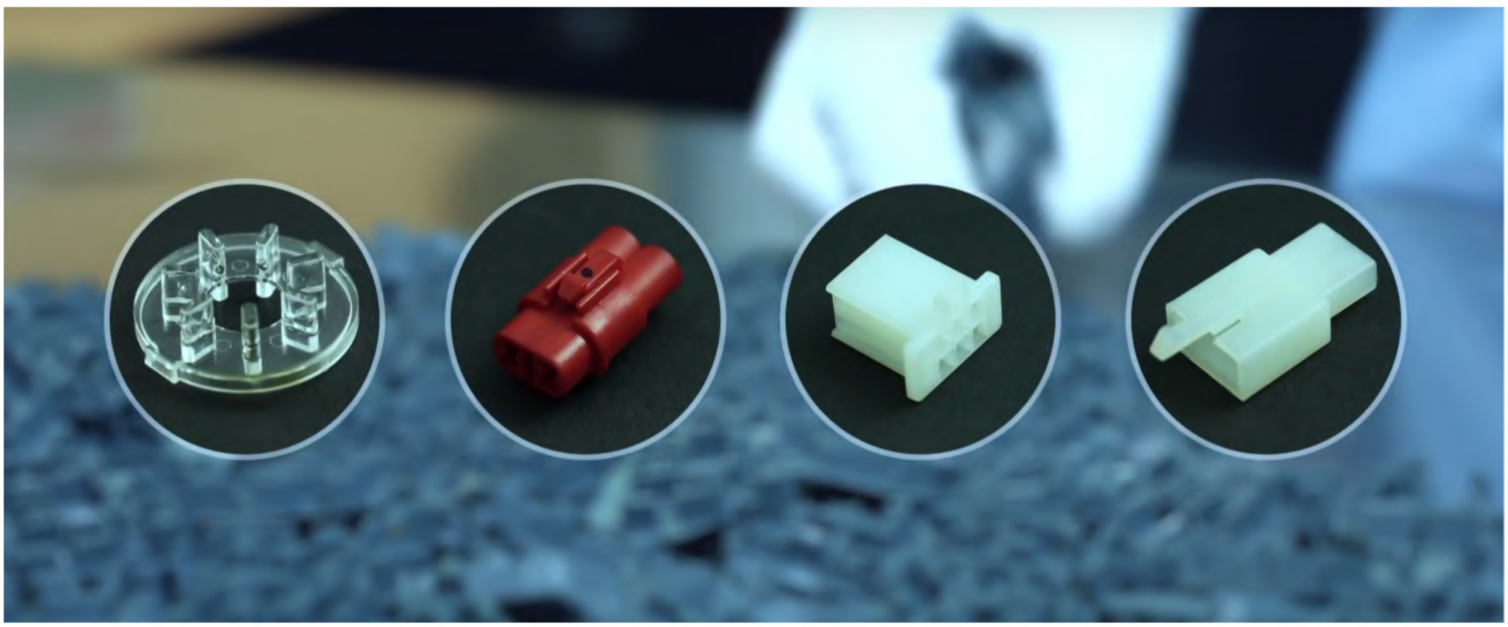
- Acrylic: 0.025 – 0.500
- Nylon: 0.030 – 0.115
- Liquid Crystal Polymer: 0.030 – 0.120
- Polyester: 0.025 – 0.125
- Polypropylene: 0.025 – 0.150
- Polyurethane: 0.080 – 0.750

These recommendations are based on years of research and development, and helps the manufacturers to get top quality results.

#2. Right radius at all corners

There are various pointers that need to be considered in this regard. If there are designs with corners, these will need to accommodate large radii. The manufacturing of the parts may be affected if the corners are sharp, and so this becomes an important aspect that needs to be taken into consideration. The corners attached between the surfaces and bosses also need careful scrutiny because these are overlooked most of the times.

Every possibility of high stress concentration needs to be eliminated and so it is crucial to set the radius based on the thickness of a part. As a general guideline, it is recommended to keep the thickness at the corner in the range of 0.9 times to 1.2 times the nominal thickness of a part.



#3. Part cost needs to come down

This is something that every client and manufacturer would want isn't it? Well, how exactly do you do that? When it comes to injection molding, there are various ways in which this can be accomplished. However, the approach towards the overall process needs to be streamlined so that the part cost can be brought down and good quality can be maintained at the same time. Here are some of the efficient ways in which this can be accomplished:

- The least thickness compliant needs to be used with the process, product design requirements, and the material.
- Using the least thickness compliant will ensure rapid cooling, minimum short weight, short cycle times, and other such practical benefits.
- Following this recommendation will ensure that the overall part cost comes down and this will have a bearing on the complete project cost as well.

These small yet effective changes when implied will ensure that the manufacture as well as the client saves on the overall project cost.

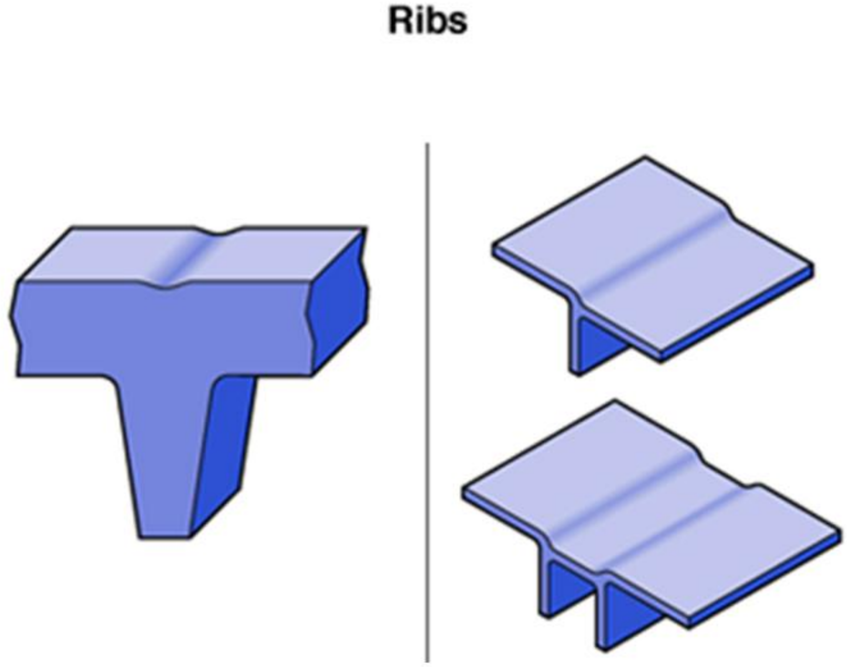
#4. Determining the right location for the gate

Although the general guidelines are to have uniform thickness in the plastic but still there are some variations required in various designs. When there are situations that require variations in the designs, it is crucial to have proper gate location in order to determine the overall success of the project.

The decision varies depending upon the project and material being worked, but in most of the cases, the experts recommend designs that have gate at the location where melt enters the thickest section of the cavity so that it can flow out of the narrow region. Taking extreme care and getting the right recommendation will ensure a successful injection molding project.

#5. The inclusion of ribs

Plastic is stiff in nature and comes with many other benefits in the injection molding process. Considering this nature, the plastic including ribs in a design is often times recommended that adds to the bending stiffness of the plastic. Ribs are inexpensive and convenient feature that benefit the manufacturer and designer at the same time. There are numerous benefits that come along with using ribs in plastic injection molding.



However, there are also certain important considerations that designers need to keep in mind. One of the main considerations that the designer needs to keep in mind is the wall thickness. If deep and thick ribs are used in the molding then there are increased chances of filling problems and sink marks. The rib thickness should never increase wall thickness because this will only result in more problems. Usually, the basic principle to keep in mind is to have a base with 0.6 times the nominal wall thickness compared to the part that is being molded. If proper rib is not included in the project, there are chances

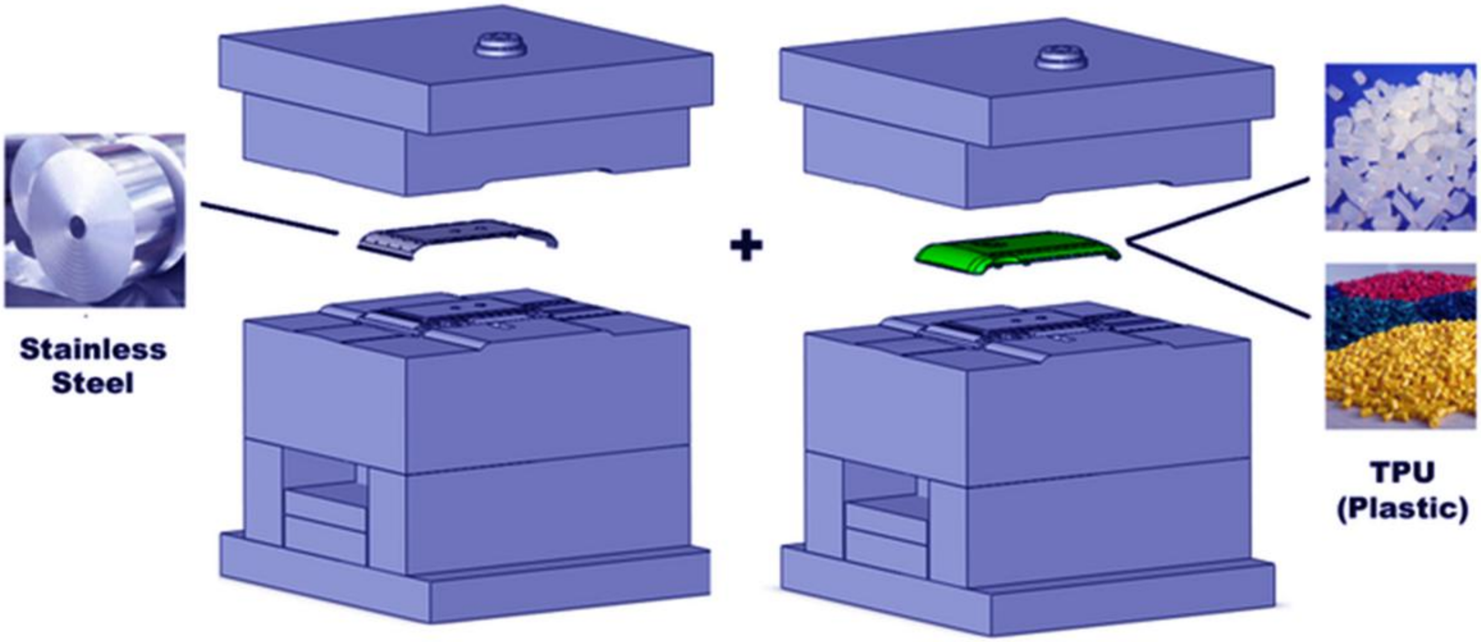
#6. Mold Draft needs to be considered

During the course of its removal from the mold, the plastic heavily relies on the mold draft. As a result, plastic parts need to be designed with a taper or draft as it is commonly known in this industry. This needs to be designed perfectly in the direction where the mold moves. The type of draft depends on how a specific feature is formed in the mold. The features that are formed by pocket or blind holes such as ribs, bosses, or posts should taper thinner as they extend into the mold. If the steel separates from surface before ejection then the surfaces formed by the slides may not need draft. Some of the other guidelines to consider in this process are:

- All the surfaces need to be drafted parallel to the mold separation.
- For every 0.001 inch of texture depth it should be a standard rule to use one degree of draft plus an additional degree of draft.
- In order to retain uniform wall thickness and assist ejection it is vital to angle walls and other such attributes that are formed in both the mold halves.
- It is crucial to use draft angle of at least one-half degree for most of the materials being used in the injection molding.

#7 Use of inserts

The parts when used in areas such as machines or other places where fixtures are required, the inserts are used in them. The inserts are robust because most of the times these are made of brass. This makes it very easy for assembling and disassembling of the products or parts. Some of the methods used to install inserts are:



- **Thermal Insertion:**

This is a much easier process than it sounds. In this process the insert is placed in the required location and then an instrument is used to heat these. After heating the insert and melting plastic, it is pressed in the plastic and fixed. Once the plastic cools around the insert, it shrinks and captures insert in a right place. Most of the companies opt for this process as it is simple and cost effective.

- **Molding in:**

In this process, the core pins are used to hold inserts in place during the molding cycle. This might be a slow process because in this, the inserts need to be hand loaded, but on the other hand, the secondary operations can be eliminated. However, if there is a big project, then automatic loading process can also be introduced but it only increases the overall cost of the project.

- **Ultrasonic insertion:**

In this process, an ultrasonic transducer is used to vibrate the insert and put it in place. This is called as horn which is mounted in the ultrasonic device and is also an effective process that is used by many companies. In this process, the ultrasonic energy is converted into thermal energy by vibrating the insert and this helps the insert to melt into the hole. This insertion technique comes with short cycle times and can be done rapidly with low residual stress.

A proper insert molding process is all about implementing the right processes and using the appropriate guidelines for the same.

Injection molding is a domain that needs a lot of quality assurance and implementation of specific guidelines. As a client, it is crucial to make sure that only the manufacturer who implements all the above mentioned guidelines is given the project. These guidelines will help the companies to come up with efficient injection molding with affordable price.

EIGEN

Your Partner for Precision

Thank You

Eigen Engineering
No 408, 4th Main, 11th Cross,
4th Phase, Peenya Industrial Area,
Bangalore-560058 India.
Phone: (+91) 8040338899
Ext: 316 / 312 / 311 / 318
Email: info@eigenengineering.com
Fax: (+91) (80) 41272189