

FOUR CAVITY MOLD FOR U CLIP

Tanishq Chandrakant Patil¹, Manasi Nitin Pawar², Shivraj Amit Thorat³, Dishant Mohan Patil⁴
Prof Y.M.Halde

1,2,3,4 Bachelor Student, Department of Mechanical Engineering Rajashri Shahu Maharaj Polytechnic,
Nashik

ABSTRACT

Injection molded components are consistently designed to minimize the design and manufacturing information content of the enterprise system. The resulting designs, however, are extremely complex and frequently exhibit coupling between multiple qualities attributes. Axiomatic design principles were applied to the injection molding process to add control parameters that enable the spatial and dynamic decoupling of multiple quality attributes in the molded part. There are three major benefits of the process redesign effort. First, closed loop pressure control has enabled tight coupling between the mass and momentum equations.

Keyword: *Injection molded components are consistently designed to minimize the design and manufacturing information content of the enterprise system.*

1. Introduction

Injection molding is used to produce thin-walled plastic parts for a wide variety of applications, one of the most common being plastic housings. Plastic housing is a thin-walled enclosure, often requiring many ribs and bosses on the interior. These housings are used in a variety of products including household appliances, consumer electronics, power tools, and as automotive dashboards. Other common thin-walled products include different types of open containers, such as buckets. Injection molding is also used to produce several everyday items such as toothbrushes or small plastic toys. Many medical devices, including valves and syringes, are manufactured using injection molding as well.

1.1 Problem Statement

To design and analysis the multi-cavity injection mould tool for the component Bulb holder. Which is use in the house hold appliance Solid works 2021 software is used for both design and analysis. The design of component is tough part in the dissertation because of complex geometry shape. And analysis is done to find the filling time, cooling time and volumetric shrinkage to for check the time taken for filling and cooling to avoid defects on cavity.

1.2 Objectives

- Study of component has been made carefully as an existing Two-plate injection mold for four cavities with edge gate was producing components with defects.
- The study of component has been done to design the tool as simplest as possible by using SolidWorks 2021 software.
- The study of specified material has been done, to know its physical and mechanical properties associated with molding material and molding characteristics that influence tool design.
- Mold flow analysis has been carried out for the component in order to achieve good quality mold before molding by using solid works 2021.

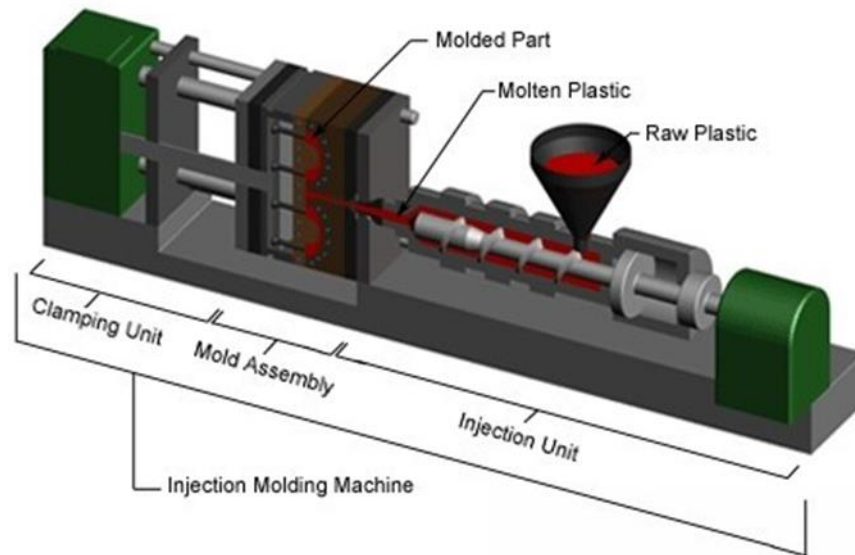


Fig -1: Injection molding overview

2: Components

The process cycle for injection molding is very short, typically between 2 seconds and 2 minutes, and consists of the following four stages:

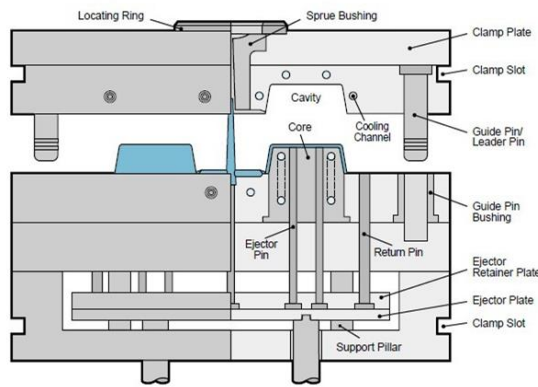
- Clamping
- Injection
- Cooling
- Ejection

Material	Specific Gravity	Melting Point (°F)
Epoxy	1.12 to 1.24	248
Phenolic	1.34 to 1.95	248
Nylon	1.01 to 1.15	381 to 509
Polyethylene	0.91 to 0.965	230 to 243
Polystyrene	1.04 to 1.07	338

Table -1 Power Requirements

2.2: Detailed Components

- I. Clamp Plates
- II. Nozzle/Sprue Bushing
- III. Feed System
- IV. Cavities



Cavity (A) side (stationary side)

- Locating Ring
- Sprue (Bushing)
- Clamp Plate
- Clamp Slot
- Guide Pin
- Cooling Channel (Water Lines)
- Cavity

Core (B) side (moving side)

- Core
- Ejector Pin
- Guide Pin
- Ejector Plate
- Ejector Retaining Plate
- Support Pillar
- Clamp Slot

Fig-2: Detailed Components

3. Mold Design

In addition to runners and gates, there are many other design issues that must be considered in the design of the molds. Firstly, the mold must allow the molten plastic to flow easily into all of the cavities. Equally important is the removal of the solidified part from the mold, so a draft angle must be applied to the mold walls. The design of the mold must also accommodate any complex features on the part, such as undercuts or threads, which will require additional mold pieces. Most of these devices slide into the part cavity through the side of the mold, and are therefore known as slides, or side-actions. The most common type of side-action is a side-core which enables an external undercut to be molded. Other devices enter through the end of the mold along the parting direction, such as internal core lifters, which can form an internal undercut. To mold threads into the part, an unscrewing device is needed, which can rotate out of the mold after the threads have been formed.

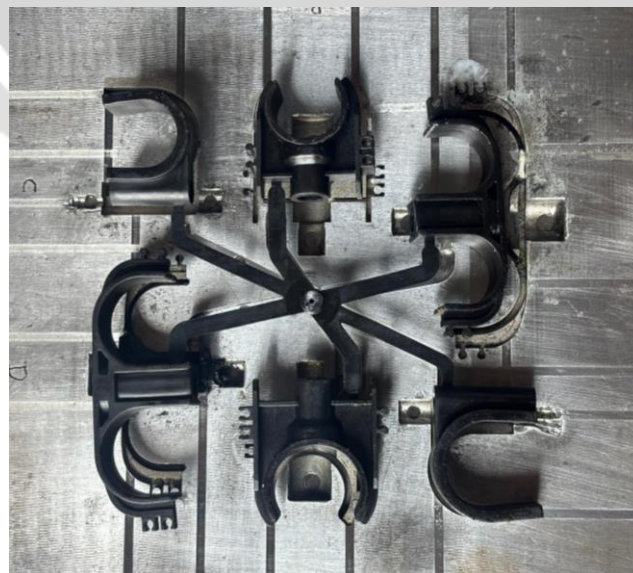


Fig-3: U clip mold

4. Conclusion

Injection moulding is a highly productive and efficient method in fabricating plastic products. The injection moulding cycle involves an injection of molten plastic into a Mold, then cooling it to form the solidified part. The Mold is customized tooling that produces identical parts consistently. Injection molding is a highly productive and efficient method in fabricating plastic products. The injection molding cycle involves an injection of molten plastic into a mold, then cooling it to form the solidified part. The mold is customized tooling that produces identical parts consistently.

5. Application

- Packaging
- Consumer goods
- Medical devices
- Electronics & telecommunications
- Mechanical parts (including gears)
- Most other common plastic products available today

5. Advantages

- It allows for complex geometries with tight tolerances. Injection molding allows for large volumes of uniform, complex parts.
- It's compatible with a wide range of materials and colours.
- It's very efficient.
- It offers high repeatability and reliability.
- You can reuse material.

7. Future work

Some of the new tendencies and technology in injection molding are the electric injection machines and the gas assisted injection molding. The electric machines have several advantages over the old design of the conventional injection machine. It runs silent, its operating cost is less, and they are more accurate and stable.

8. REFERENCES

1] A Brief Review on injection moulding manufacturing process

<https://www.sciencedirect.com/science/article/abs/pii/S2214785317301645>

2] Recent methods for optimization of plastic injection moulding process

https://www.researchgate.net/publication/50346402_Recent_methods_for_optimization_of_plastic_injection_moulding_process-a_retrospective_and_literature_review

3] Advanced Injection Moulding Methods

<https://www.mdpi.com/1996-1944/16/17/5802>