iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: V Month of publication: May 2018

DOI: http://doi.org/10.22214/ijraset.2018.5035

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

Design and Manufacturing of Thermoplastic Fuel Clamp

Nagendra Akula[1], Dr.K.Rajagopal[2]
[1] Asst. Prof. Dept. Of Mechanical Engineering, St. Peters Engineering College, Research Scholar ,JNTUK)
[2]Professor of Mechanical Engineering, SKD University, Ananthapuramu)

Abstract: The process of Tool & Die Making involves a series of direct stages which completes in a proper planning and proper thinking and co-ordination must result in a successive tool. This report gives a vision of 'ART IN TOOL & DIE MAKING' Designing is used to review the product concept for manufacturing. Dividing the total processing into discrete module like raw material selection, machining, job follow up, assembly and final tryout. This project report has been prepared with an outlook to import optimum information about a tool in a more easily grasping in a short span of time. It gives knowledge of the tool making techniques, difficulties involved and the skill of a tool maker. The aim of this project report is to explain each subject from foundation to the finish of the project in a simple and clear manner, with the help of various tools (single point cutting tool, drill bits, end milling cutters, ball nose cutters, radius end mills, sparking in EDM etc.) used for manufacturing Dies.
Keywords - Jigs, Fixtures, Clamp, Tools, Dies, Cutting tools, Drilling.

I. INTRODUCTION

A. Objective of the Project

This project was carried out at ACME TOOLING.

1) The project Objective is to design an injection mould for ITW (FUEL CLAMP).
2) To manufacture and assemble the mould.
3) To make production of the component.

B. Injection Moulding

Mould is a tool, designed to form the component with desired shape and size. The type of material to be moulded and the shape of the component are the major factor governing to choice the mould.Plastic moulding is especially thermoplastic items may be produced by injection moulding method. Injection moulding differs from compression moulding is that the plastic material is formed as a fluid in a separate chamber or barrel outside the mould is then forced into the mould cavity by external pressure. The mould is cooled and split, the two halves been locked by pressure to resist injection force during the moulding and opening automatically when the mould cavity has been filled by the molten plastic. The cooled hardened article is then ejected from the mould and the process repeated.

The compound is Derlin100p material. It has critical profiles & critical dimensions and it has closed tolerance so it is easy for injection moulding.

There are various types in injection moulding. They are

1) Two plate mould.
2) Three plate mould.
3) Hot runner mould

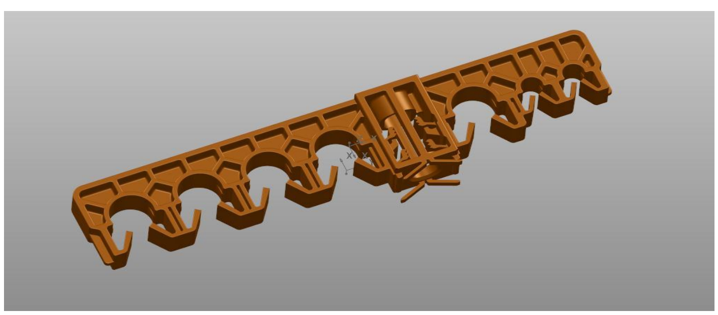


Figure 1 Injection Moulding

C. Component Details

It is an important factor to be considered that before analyzing the design of a component. The component has to be studied well in a perfect way knowing about its material.

Table1: Component details

COMPONENT NAME	FUEL CLAMP
MATERIAL	DERLIN 100P
SHRINKAGE	2.2%
GATE TYPE	SUB MARINE
NO. OF CAVITIES	2
WEIGHT OF COMPONENT	22GMS
SHOT WEIGHT.	55GMS
MOULD WEIGHT	300KGS
TYPE OF EJECTION	PIN EJECTION

II. MATERIAL PROPERTIES OF DERLIN

The resins in the Delrin® 100 acetalhomo polymer series are distinguished by their high molecular weight relative to other acetal grades. This identifying feature of these high performance acetalhomopolymers, along with their natural high crystallinity, yields materials that have:

A. Toughness, high impact strength, high resistance to repeated impact, and high elongation without the need for impact modifiers.
B. High mechanical strength and rigidity, without the need for fillers/reinforcements or other modifications.
C. Outstanding creep resistance and long-term fatigue endurance.

These features of high mechanical strength and toughness can be seen in this overlay of stress-strain curves of high viscosity Delrin® acetalhomopolymers and standard acetal copolymers of similar viscosity (Melt Flow Rate of 2.5 g/10 min at 190°C). Delrin® 100 and 100P have very high Tensile Strength and Strain at Break, significantly higher than the standard copolymer.

Delrin® acetalhomopolymers are readily processed with a variety of molding machines and tooling configurations. The good flow characteristics, fast cycling, and thermal stability of Delrin® grades allow for these materials to be easily molded into complex shapes for parts in demanding acetal applications.

III. MOULD

Mould is a hollow from or cavity into which molten plastic material is forced to give the shape of the required component. The term generally refers to the whole assembly of parts which go to make up the section of the injection moulding equipment in which plastic components are moulded.

A. Types of Moulds

1) Compression mouldin
2) Transfer moulding
3) Blow mouldin
4) Extrusion mouldin
5) Injection moulding

The present time is considered to be the plastic age. Where an unlimited number of plastic are being made available which in turns broadens the avenues for a wide variety of applications.

All plastics can be made to flow by the application of heat and pressure to assume the shape of the mould. One of the most outstanding features of plastics is, ease with which they can be processed.

The main function of an injection moulding machine is to make perform into solid and rigid. Mainly compression moulding machine can be divided into 4 units.

a) *Injection Unit:* Injection unit mainly consists of fixed and movable platen with hydraulic cylinders with heating unit and oil tank

b) *Clamping Unit:* The main function of clamping unit is to hold the platen under pressure to Overcome Injection pressure

c) *Locking Unit:*There are mainly three types of die locking:
a) Hydraulic plunger typ
b) Mechanical typ
c) Hydro toggle (this is the most commonly used one)Ejector Unit: It consists of knockout plate mounted behind the moving platen. This knockouts plate houses the knockout rods. When the die is opened, the knockout rod comes into contact with ejector unit forward which in turn ejects the components. There are four basic types of injection mounding equipment's in use today.

Problem	Molding condition	Molding compound	Mould
1.Breakage on rejection	-Soft material -Temperature too low -Cure time is too short -Hard material -Mould temperature too high		-Insufficient draft -Uneven ejection -Undercuts
2.Thick flash	-Temperature too high -Too much material -Pressure too high -Press closure too slow	-Too stiff -Too fast curing	-Mould not shutting properly
3.Execessiveflash tightness in cavity	-Too much material	-Too easy in flow	-Flash clearance too large
4.Short shot	-Temperature too high -Insufficient charge weight -Pressure too low -Closing speed too fast -Fast curing	-Too stiff in flow	-Diameter of pins too small -Pins not fully returned
5.Sink mark	-Insufficient charge weight -Pressure too low	-Too easy inflow	-Flash clearance too large
6.Cracks	-Very rare		-Uneven ejection -Undercuts, Rough surface

IV. TOOL DESIGN CALCULATIONS AND DRAWINGS

A. *Design Consideration*

The tool design must always perform the specified functions. It must meet various requirements and have good tool life at a minimum cost. To meet various requirement, the designer as to design the tool in a methodical way.

Before designing this mould the following points were taken into consideration.

1) Machine dat
2) Production Dat
3) Number of Impressio
4) Availability of a machine
5) Component material
6) Type of too
7) Availability of machines for manufacturing the tools

8) Grade of tool material
9) Standard design data
10) Cost of the tool

B. Tool Design Analysis

Design is a specialized phase in engineering field. In mould engineering it is totally divided into two wings as product design and tool &die design. In tool and mould manufacturing machine cost and die cost are too high because high precision is necessary for the quality and uniformity of the parts from the accuracy required for the tool and mould parts and replaceable elements. A number of preliminary factors must be considered before starting the actual tool design and decision on the method.

1) Type of mould and construction.
2) Number of cavities.
3) Extraction of components
4) Gate, runners, type of Feed & flow
5) Split and side core
6) Wear and tear of elements.
7) Analysis of possible problems & solutions
8) Selection of moulding machines.
9) venting

C. Extraction of Component (Ejection)

The term ejection means ejecting the component out of the mould when the moulding component strings and stick to the core and it becomes difficult to remove the component from the mould. So ejections are provided. This type of ejections depends on shapes and size of the component. For this mould pull back ejection is adopted.

D. Feed System

Feed system consists of sprue, runner and gate.

1) *Sprue:*-In the injection moulding process the plastic material is delivered to the nozzle of the machine as a melt. It is then transferred to the impression through a passage. The material in this passage is termed as sprue.
2) *Runner:*-The runner is a channel machine into the mould plate to connect the sprue with the entrance gate to the impression.
3) *Gate:*-It is a channel connecting to the runner with the impression. It has small cross section area.
4) *Venting:*- When the plastic material enters cavity air inside the mould will trapped, result the mould defects venting is the way which allows the air to escape freely. Vent is a shallow slot having 0.05 depth and 3.5mm in width.

E. Injection Mould Features

The following factors were considered while designing this mould

1) Injection Syste
2) Parting Line
3) Heating Syste
4) Ejection System
5) Core and cavity layout

Ejector pin height = Ejector Back Plate + Spacer+ Core Plate+ Insert Height
=22 Gms

V. SELECTION OF RAW MATERIALS:

Injection mould die material must be resistant to thermal shock, softening and corrosion at high temperature. The performance of die material is directly related to the injection temperature of the molten material.

To guarantee the successful operation of an injection mould, the die steel should have the following characteristics.

High compressive strength combined with sufficient toughness.

Sufficient strength & hardness to resist deformation.

Low-Coefficient of thermal expansion

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

Good wear resisting property.
Corrosion resistance
Good mach inability
Low or no distortion during heat treatment
Structural soundness & uniformity
Good polish ability
Steels containing alloying elements that enable it to heat treated to obtain desirable characteristic such as strength, toughness, wear resistance etc. can be referred to as tool steel.

A. Composition

1) Carbon 0.5-0.6
2) Silicon 0.1-0.35%
3) Manganese 0.5-0.8
4) Nickel 1.25-1.75
5) Chromium 0.5-0.8
6) Molybdenum 0.25-0.35%

B. Methods of Inspection

Inspection means the evaluation of the product to see whether it fulfills the quality requirements and then decides an acception, rejection and rework to achieve quality and perfection in the part components. It is necessary to inspect the tool parts at all stages.

1) *Purpose of inspection*
   a) To collect information regarding the performance of the product with established standards.
   b) To sort out poor quality manufactured products and thus maintain the standard.
   c) To establish and increase the reputation by protecting consumers from receiving bad products.
2) *Importance of inspection*
   a) Inspection traces defects in raw material, flaws in process, and those arising in production due to defective machinery or process.
   b) Inspection avoids further work on non-finish parts already detected as spoiled.
   c) Inspection ensures that quality of goods supplied to customers is up to mark. There are 3 basic areas of inspection: priority to dispatch. Aim is to prevent defective products from being shipped to customers.

C. Injection Unit Cycle Time

Table2: Injection cycle time

Sl No.	operations	Time/seconds
01.	Mold closing	4 seconds
02.	Injection Time	8 seconds
03.	Cooling time	10 seconds
04.	Hold on time	15 seconds
05.	Mold opening	5 seconds
06.	Ejection	2 seconds

Total: 44 seconds.

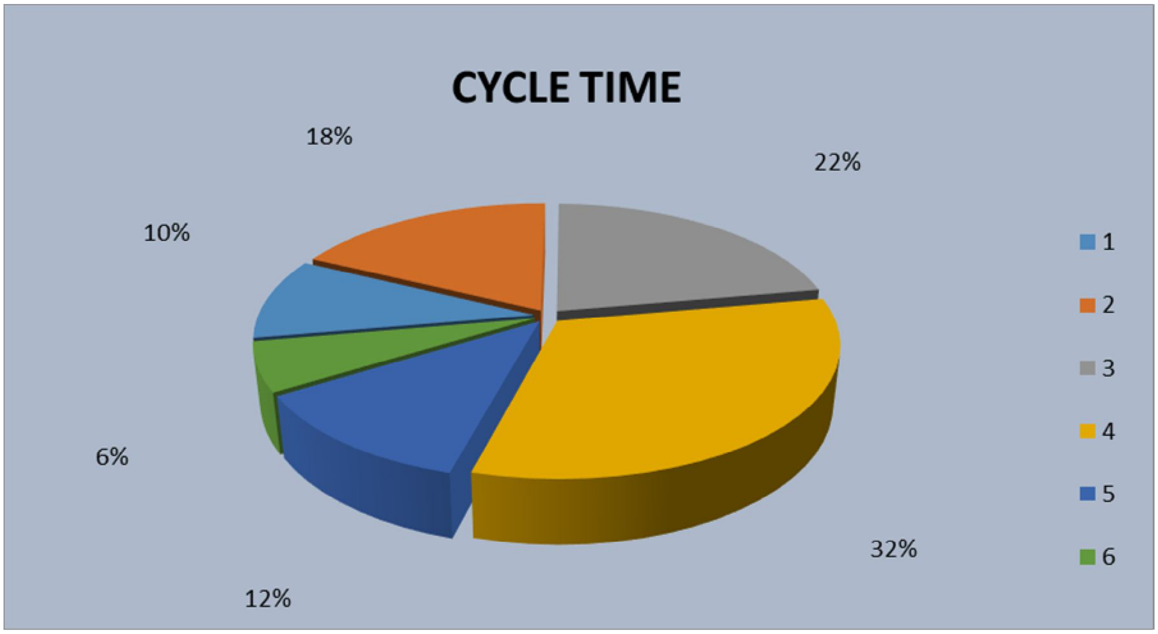


FIGURE-2 Pie Chart for Cycle Time

Cost Analysis
Calculation:

Actual cost = Total Machine Cost + Heat Treatment Cost
= 81300 + 6810
= 88110

Prime Cost = Raw Material + Machining Cost + Heat Treatment Cost + Standard Item Cost + Try Out Cost + Standard Mould Base + Design Cost
= 9496 + 81300 + 6810 + 25000 + 11500 + 50950 + 42000
= 2, 27,056

Design Cost (15% of Prime Cost) = 34058.4
Over Head Cost (30% of Prime Cost) = 68116.8
Risk factor (20% of prime cost) = 45411.2
Profit (20% of prime cost) = 45411.2

Total cost = Prime cost + Design cost + Over heads + Risk factor + Profit
= 2,27,056 + 42000 + 68116.8 + 45411.2 + 45411.2
= 4,27995.2

VI. RESULTS & DISCUSSION

A. Injection moulding is an extremely useful tool for mass-producing polymer parts.
B. All industries are in the situation to improve their productivity.
C. Every company is facing many challenging problems to produce better production rate.
D. Manufacturers should be able to identify even the small factors affecting production growth.
E. Clear identification of problem at the right time will help to increase quality as well as productivity rate.
F. Another benefit that was realized is to give them the chance to know what best techniques they can apply which will improve their performance.
G. If data are not properly collected then the resulting will not be meaningful.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

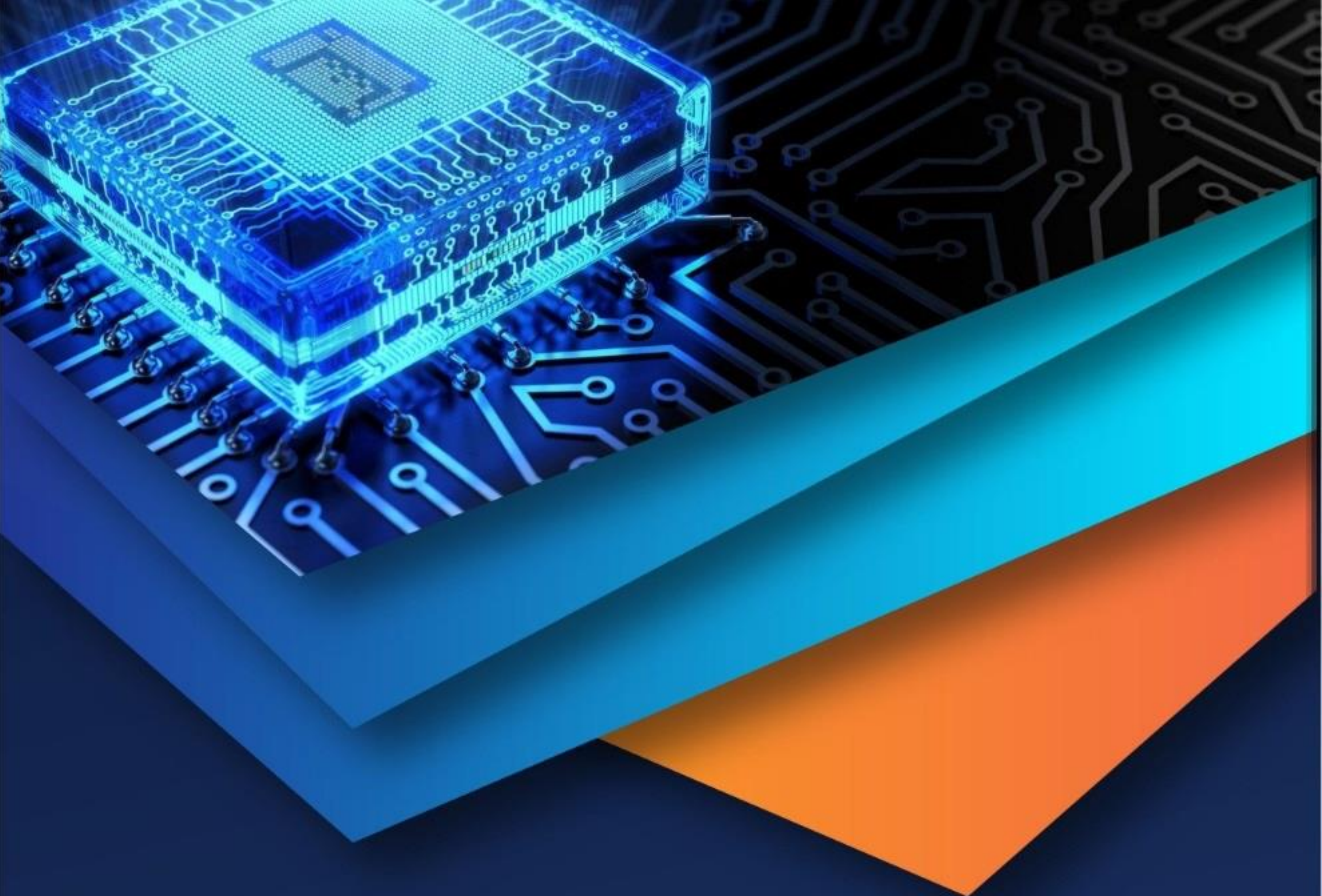
VII. CONCLUSION

Plastics are increasingly used in today's world. Injection moulding is one of the most common methods used to process plastics. We have to consider various factors to produce a defect free and economical plastic component during the design stage itself. Injection mould design for parts is done using Pro-E, auto cad and simple software. The different processing parameters and machine selection is according to the design calculations done.

The mould base and inserts are being manufactured at the manufacturing facilities available with ACME TOOLING. Production of components is to be done at the in-house customer's end. Hence the mould is supplied to the in-house customer after acceptance of the sample by them.

REFERENCES

[1] Henriksen, E. K. Jig and fixture design manual, 1973 (Industrial Press Inc, New York)
[2] Gouldson, C. J. Principles and concepts of fixturing for NC machining centers. In Jigs and fixtures (Ed. Boyes, W.), 1982 (Society of Manufacturing Engineers, Dearborn, Michigan)
[3] R. Hunter Alarcon, J. Rios Chueco, J.M. Perez Garcia, A. Vizan Idoipe, "Fixture knowledge model development and implementation based on a functional design approach", Robotics and Computer-Integrated Manufacturing, 26 (2010), pp. 56-66
[4] Advances in integrated product design and manufacturing, P. H. Cohen, S. B. Joshi, American Society of Mechanical Engineers. Winter Meeting, American Society of Mechanical Engineers.
[5] Joshi, "Jigs and Fixtures", Tata McGraw-Hill Education, 2001
[6] K. Venkataraman, "Design of Jigs, Fixtures and Press Tools", John Wiley & Sons, 2015
[7] Edward Hoffman, "Jig and Fixture Design", Cengage Learning, 2012



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY