Session 2019-23

Project Supervisor

Engr. Fawad Yousef Malik

Group Members

Mubashir Asif UET/SCET-19F-ME-039

Shahzaib Saleem UET/SCET-19F-ME-013

Muhammad Ahsan UET/SCET-19F-ME-019

DEPARTMENT OF MECHANICAL ENGINEERING SWEDISH COLLEGE OF ENGINEERING AND TECHNOLOGY, WAH CANTT

(Affiliated with University of Engineering & Technology Taxila)

August 2023

A Thesis Submitted for Partial Fulfillment of The Requirements for the Degree

Of

Bachelor of Science in Mechanical Engineering

Project Supervisor

Engr. Fawad Yousaf Malik

Project Coordinator

Engr. Hassan Kazmi

Dr. Liaqat Ali Najmi

HOD

Mechanical Engineering Department

DEPARTMENT OF MECHANICAL ENGINEERING SWEDISH COLLEGE OF ENGINEERING AND TECHNOLOGY, WAH CANTT

(Affiliated with the University of Engineering & Technology, Taxila)

August 2023

Abstract

PVC pipe clips are essential components used in various industries to securely fasten and organize PVC pipes. This project aimed to develop a cost-effective and efficient manufacturing process to produce high-quality PVC pipe clips with consistent dimensions and superior performance. This research initially focused on understanding the requirements and specifications of the PVC pipe clips and analyzing existing molds used for similar purposes. design and fabrication of injection molding molds play a crucial role in modern manufacturing processes, enabling the mass production of intricate plastic components used in various industries. My research study presents a comprehensive investigation into the design and fabrication of an injection molding mold, focusing on optimizing its performance, efficiency, and quality. So, the initial phase of the study involved an in-depth analysis of the requirements and specifications of the target plastic component. Through computer-aided design (CAD) software, a precise and innovative mold design was developed, considering factors like material compatibility, part geometry, and mold complexity. Special attention was given to the selection of appropriate materials for the mold to ensure durability and longevity under the demanding conditions of the injection molding process. The fabrication process employed advanced machining techniques, such as computer numerical control (CNC) machining, to manufacture the mold components with high precision and accuracy. The use of modern manufacturing technologies allowed for the rapid production of complex mold geometries, reducing lead times and enhancing overall production efficiency. Investigated various aspects of mold design and fabrication, including cooling system optimization, venting solutions, and surface treatments, to minimize defects and improve the quality of molded parts. The researchers also explored the integration of conformal cooling channels to enhance heat dissipation outcomes of this research contribute to the advancement of injection molding technology by providing insights into the best practices for mold design and fabrication. The developed methodologies and findings can serve as valuable guidelines for manufacturers seeking to optimize their production processes, improve product quality, and reduce production costs and reduce cycle times.

Declaration

We hereby declare that the project work entitled "Designing & Fabrication of Die for Injection Molding Machine" is completed and submitted to the SCET Wah, is a record of work done by us under the guidance of project supervisor Engr. Fawad Yousaf Malik and this project work is submitted in partial fulfillment of the requirements for the award of the degree of BSc in Mechanical Engineering. The results embodied in this project have not been submitted to any other University or Institute for the award of any degree or diploma.

(Fawad Yousaf Malik)

Project Supervisor

Group Members

(Mubashir Asif)

UET/SCET-19F-ME-039

(Shahzaib Saleem)

UET/SCET-19F-ME-013

(Muhammad Ahsan)

UET/SCET-19F-ME-019

Acknowledgment

In The Name of Allah, The Most Gracious and the Most Merciful. All praise and thanks are to Almighty Allah, the most merciful and compassionate, who has granted us the opportunity, the ability, and the resources to complete this work.

We would like to express our sincere gratitude to our parents and family for their unwavering support, love, and encouragement throughout my academic journey.

We would also like to thank my supervisor Engr. Fawad Yousef Malik for their valuable guidance, constructive feedback, and patience during the development of this Project. His expertise and dedication have been instrumental in shaping my research and enabling me to produce quality output.

We would also like to express our sincere gratitude to our HOD Dr. Liaquat Ali Najmi, Engr. Engr. Muhammad Naeem, Engr. Hasan Kazmi and all the Faculty members for guiding us throughout.

Dedication

Dedicated to all God Almighty my creator, my strong pillar, my source of inspiration, wisdom, knowledge, and understanding. He has been the source of my strength throughout this program. We also dedicate this work to our parents who have encouraged us all the way and whose encouragement has made sure that we give it all it takes to finish that which I have started. Thank you. Our love for you all can never be quantified. God bless you.

Table of Contents

No table of figures entries found.

No list of tables entries found.

Chapter 1

INTRODUCTION

1.1 Introduction

Injection molding is a widely used manufacturing process for plastic components, and the mold for the injection molding machine is its core element. This introduction highlights the mold's significance and key factors in design and operation. The mold shapes molten plastic into the desired form, consisting of cavity and core halves precisely machined to create the product's negative impression. Proper mold design considers geometry, materials, cooling systems, and gate placement to ensure smooth filling and minimal defects. Material selection involves using steel or aluminum to withstand high pressures and temperatures. Different mold types accommodate varying production needs, from single-cavity molds for low volume to multi-cavity and family molds for higher output. Regular maintenance and innovative technologies, such as CAD/CAM, contribute to enhancing mold performance and prolonging its lifespan, enabling the production of high-quality plastic products efficiently.

1.2 Aims and Objectives.

To design and manufacture the PVC drainage cover die.

For making of PVC pipe holding clip die will be designed by first we have to know the peak pressure and temperature of the injection molding machine on the basis of temperature and pressure the material selection will be done by testing all calculation in Ansys software so that in result all the percentages and composition of materials will be clarified with precautions.

1.3 Historical Background

Die is believed to have had its humble birth sometime during the middle of the 19th century. According to records, the first die equipment was invented in 1838 with the sole purpose of manufacturing parts for the moveable type machines responsible for the printing industry. Now a day the die is installed in the injection molding machine so that that speeds up the process and no man power is used during this process. The earliest ASTM standards for PVC sanitary sewer pipes/drains were approved in 1972. However, the use of PVC pipe in both drainage and sewer applications in North America dates back to the 1952. In the 1950s dominated the North American sanitary sewer market.

1.4 Design Components

The design components of an injection molding machine mold can vary depending on the specific requirements of the product being manufactured. However, here are some essential design components commonly found in injection molding machine molds:

1.4.1 Cavity and Core:

These are the main components that form the shape of the final product. The cavity is the female part of the mold, and the core is the male part. When the mold is closed, the cavity and core come together to create the desired shape of the product.

1.4.2 Runner System:

The runner system is responsible for guiding the molten plastic from the injection nozzle into the cavities.

It consists of channels and gates that distribute the plastic evenly to each cavity.

1.4.3 Gating System:

The gating system includes the sprue, runners, and gates that direct the flow of molten plastic from the injection machine nozzle to the cavity. The design of the gating system affects the flow, cooling, and quality of the molded part.

1.4.4 Ejector System:

The ejector system is used to push the finished part out of the mold after it has cooled and solidified. It typically consists of ejector pins or plates that move the part away from the core.

1.4.5 Cooling System:

Efficient cooling is crucial in the injection molding process to ensure proper solidification and minimize cycle time. Cooling channels are incorporated into the mold to remove heat from the molten plastic and the mold itself.

1.4.6 Venting System:

To prevent air traps and ensure complete filling of the mold, vents are included in the design. These vents allow air and gases to escape during the injection process.

1.4.7 Guiding and Alignment Mechanisms:

These components ensure precise alignment and proper closure of the mold during the injection process.

They help maintain the correct position of the cavity and core.

1.4.8 Ejector Pin Retainers:

These components hold the ejector pins in place during the molding process and allow for easy replacement if needed.

1.4.9 Sprue Puller System:

The sprue puller is a mechanical device that helps to separate the sprue (the excess material connected to the runners) from the molded part.

1.4.10 Heating System:

Some molds, especially for certain types of materials, may include a heating system to aid in maintaining proper mold temperature and optimize the molding process.

1.4.11 Mold Base:

The mold base is the main structural support for all the components and provides the interface to the injection molding machine.

1.4.12 Ventilation System:

The ventilation system allows for proper air circulation during cooling and can also aid in exhaust management.

1.4.13 Overflows and Parting Lines:

These are essential features in the mold design that account for material shrinkage during cooling and enable easy removal of the finished part from the mold.

1.5 Methodology

1.5.1 Theoretical Studies

Acquiring dimensions-The dimensions of the existing die were acquired by manually by measuring with a scale and Vernier scale. Also, more accurate dimensions were acquired by scanning with a CMM at Accurate.

1.5.2 Experimental Setup

Creating a CAD model of existing die - By using the dimensions we obtained we could create a CAD model using software like CATIA, Creo, Solid Works, etc.

If the product is scanned by a CMM we would obtain a CMM data sheet which contains the coordinate points plotted in a 3D plane with the help of a computer integrated software like Arco software or other assistant software. By using laser scanner, we could directly generate a 3D CAD model of the product as it is scanned.

1.5.3 Material Selection for die

A suitable material is selected under the guidance of guide, research papers and support from a local injection company. The below comparison helps us to select the material for die based upon our requirements. As per our requirement.

1.5.4 Design of CAD model for new product

A design of product is modeled using a modelling software such as CATIA using the constraints obtained after measuring the dimensions of existing die and molding machine.

1.5.5 Design of CAD model for New Die

A design for new die is to be prepared by using CATIA or other more compatible software with simulation. To create a design a suitable CAD software such as CATIA, ProE, SOLIDWORKS, etc can be used. For our designing we have used the designing software.

The shape of the PVC water drainage net was designed by using an existing pvc water drain net available in market.

Similarly, the new die was designed by using the dimensions of the existing product with slight modification to accommodate the new product design.

Also, the shape of die is designed to be rectangular to reduce cost and avoid excess machining processes.

The length of die is kept increased to accommodate the pvc water drainage net cavity and some more dimensional allowance is provided so both halves of die have greater contact surface area.

This helps in generating the pressure required to hold the halve together when screwed together with manual handle and then vertical injection is d one.

The cavity in the die is slightly larger than the designed product as we have to provide shrinkage allowance of 0.1-1 mm.

Also, draft angles at the cavity borders or the parting line is necessary to facilitate easy removal of product after cooling

1.6 Expected Results

1.6.1 Finalization of Die and Part

After successful simulation we would be certain that a part would be generated by using the new die design. After this we can move forward to manufacturing.

1.6.2 Manufacturing of Die

The die is to be manufactured by using VMC machine. If the design is cost feasible then we can also opt for a rapid prototyping process such as 3D printing or laser sintering.

1.6.3 Testing of Die

The die needs to be tested on an injection molding machine. The part which is produced after cooling of the plastic should have a proper shape and good enough surface finish. The part should not distort at removal or stick to the die surface. If this is successful, then the die is ready to be used in manufacturing. Modification (if necessary) after testing of die

After testing of die if any defects are found then we need to identify those and rectify them using suitable measures. The die should be prepared and simulated carefully before manufacturing to avoid this step as it can result in waste of time, money and material.

1.7 Utilization of Results

The product is now a day widely used in our daily life in every perspective e.g., like homes, colleges, etc.

1.8 Work Schedule Plan

The total duration of the project is 12 months which is further divided into following four phases.

• Phase 1(Collection of relevant literature and Data collection)

The estimated time for this phase is 3 months in which different literatures will be studied.

• Phase 2(Experimental Work)

Experiments will be performed on test rigs for analysis under fretting fatigue. The estimated time for this phase is 4 months.

• Phase 3(Compilation of Results)

The Estimated time for this will be 3 Months.

• Phase 4 (Data Analysis and Thesis Writeup)

The Estimated time for this will be 3 months.

Since, the work is to be documented and submitted to concerned quarters in the form of a report for evaluation, also include time slots in your project schedule for preparation and auditing of the Project Report. Organization and Management Project Work Schedule.

PHASE- I	Collection of relevant Literature & data Collection													
PHASE- II	Experimental Work													
PHASE- III	Compilation of Results													
PHASE- IV	Analysis of Data & Thesis Writeup													
Months		Aug	Sep	Oct	Nov	Dec	Jan	Feb	Mar	April	May	Jun	Jul	Aug

Chapter 2

LITERATURE REVIEW

2.1 Practical applications of taguchi method for optimization of processing parameters for plastic injection molding: a retrospective review

NgChina Fei,Nik Mizamzul Mehat, Shahrul (2013)

This paper presents a review of research in the optimization of processing parameters for injection moulding. A number of research works based on standalone Taguchi method and the integration of Taguchi method with various approaches, including numerical simulation, grey relational analysis (GRA), principal component analysis (PCA), artificial neural network (ANN), and genetic algorithm (GA), have been discussed. In a volatile and fiercely competitive global market, the practice of the trial-and-error approach which relies heavily on the experience of the moulding personnel is no longer sufficient to meet the challenges of globalization especially at the point where the disadvantages outweigh its advantages. (1)

2.2 Investigating the cutting phenomena in free-form milling using a ball-end cutting tool for die and mold manufacturing

Adriano Fagali de Souza Anselmo Eduardo Diniz, Alessandro Roger Rodrigues, Reginaldo, Teixeira Coelho (2007)

This paper presents an investigation of no planar tool-work piece interactions in free-form milling using a ball end cutting tool, a technique that is widely applied in the manufacturing of dies and molds. The influence of the cutting speed on the cutting forces, surface quality of the work piece, and chip formation was evaluated by considering the specific alterations of the contact between tool-surface along the cutting time... When the tool tip center is in the cut region, the material is removed by shearing together with plastic deformation. Such conditions increase the cutting force and surface roughness and lead to an unstable machining process, what was also confirmed by the chips collected. (2)

2.3 Productivity enhancement in dies and molds manufacturing by the use of C1 continuous tool path

M Boujelbene, A Moisan, N Tounsi, B Brenier (2004)

The challenge of die and mold milling is to achieve the specified dimensional and geometrical accuracy, and improve the surface roughness, in order to minimize polishing operations, still necessary to meet the tight tolerances required by the automotive industry and the plastic injection sector. Surface finish analysis given by 3D measurements and data processing techniques shows that microgeometrical defects generated by C1 continuous tool path are definitely less serious than those obtained by classical tool paths driven via linear interpolation. Also, polishing time saving is more than 30%. Thus, a substantial productivity enhancement is achieved in dies and molds manufacturing. (3)

2.4 Modern machining of die and mold tools. J mater process technol

Krajanik P, Kopac J (2004)

Modern production of die and mold tools is quite different in comparison with conventional machining. The basic theory of cutting process and cutting geometry is similar, but the techniques and technology is quite different. High-speed cutting (HSC) principles are not applicable with conventional machinetools. Difference between conventional and high-speed cutting velocity is analyzed on chip formation. The comparison between EDM and HSM has been made and shows great HSC benefits. Usage of modern software (CAM) optimizes tool production and helps us to save unnecessary additional machining time and costs. (4)

2.5 Investigating the cutting phenomena in free-form milling using a ball-end cutting tool for die and mold manufacturing

Adriano Fagali de Souza, Anselmo Eduardo Diniz, Alessandro Roger Rodrigues & Reginaldo Teixeira Coelho

(2014)

This paper presents an investigation of no planar tool-work piece interactions in free-form milling using a ballend cutting tool, a technique that is widely applied in the manufacturing of dies and molds. The influence of the cutting speed on the cutting forces, surface quality of the work piece, and chip formation was evaluated by considering the specific alterations of the contact between tool-surface along the cutting time. The experimental results demonstrated the negative effect of the engagement of the tool tip into the cut on machining performance. The length of this engagement depends on the tool and work piece curvature radii and stock material. When the tool tip center is in the cut region, the material is removed by shearing together with plastic deformation. Such conditions increase the cutting force and surface roughness and lead to an unstable machining process, what was also confirmed by the chips collected. (5)

2.6 Application of Taguchi method in the optimization of injection molding parameters for manufacturing products from plastic blend

S. Kamaruddin, Zahid A. Khan and S. H. Foong (2010)

This paper presents a study in which an attempt has been made to improve the quality characteristic (shrinkage) of an injection molding product (plastic tray) made from blends plastic (75% polypropylene (PP) and 25% low density polyethylene (LDPE)) by optimizing the injection molding parameters using the Taguchi method. The analysis of the results shows that the optimal combination for low shrinkage are low melting temperature, high injection pressure, low holding pressure, long holding time and long cooling time.

Using Taguchi method for design of experiment (DOE), other significant effects such as interaction among injection molding parameters are also investigated. (6)

2.7 A CAD/CAE-integrated injection mold design system for plastic products

Ivan Matin, Miodrag Hadzistevic, Janko Hodolic, Djordje Vukelic & Dejan Lukic (2012)

Mold design is a knowledge-intensive process. This paper describes a knowledge-based oriented, parametric, modular and feature-based integrated computer-aided design/computer-aided engineering (CAD/CAE) system for mold design. Development of CAx systems for numerical simulation of plastic injection molding and mold design has opened new possibilities of product analysis during the mold design. The proposed system integrates Pro/ENGINEER system with the specially developed module for the calculation of injection molding parameters, mold design, and selection of mold elements. The system interface uses parametric and CAD/CAE feature-based database to streamline the process of design, editing, and reviewing. Also presented are general structure and part of output results from the proposed CAD/CAE-integrated injection mold designs. (7)

2.8 Shear viscosity measurements of polymer melts using injection molding machine with adjustable slit die

Johanna Aho SeppoSyrjala (2011)

Rheological properties of polymer melts were studied using a slit die attached to an injection molding machine. The die has three exchangeable inserts to set the slit height to 0.75, 1.0, or 1.5 mm. The modular design eases the die cleaning, and measurements with variable slit heights enable the detection of possible wall slip. This suggests that a simple slit die construction can be used as an inexpensive rheometer to measure rheological properties of polymer melts for processing purposes. The thermomechanical history of the investigated melt in the in-line slit die experiments resembles that in actual melt processing and, therefore, determination of rheological properties under true processing conditions is possible. (8)

2.9 Quality monitoring of micro-shrinkage defects in thick-walled injection molded components

Jian-Yu Chena Jia-Xiang Zhuangb Ming-Shyan Huangb (2019)

Sink mark defects are commonly observed in thick-walled injection molded components as a result of underpacking in the hot-spot areas and an insufficient compensation of the polymer melt as the plastic begins to cool. These defects degrade the surface quality of the molded parts and are a reliable indicator of excessive volume shrinkage during the molding process. The experimental results obtained under various values of the holding pressure show that the five injection molding process quality indexes are all highly correlated with the microshrinkage. The correlation is particularly strong for the pressurebased quality indexes and hence, it is suggested that these indexes provide a viable means of predicting the formation of sink mark defects and excessive micro shrinkage in thick-walled injection molded component. (9)

2.10 A framework for analyzing energy efficient injection-molding die design

J. Mattis; P. Sheng; W. DiScipio; K. Leong (1996)

Significant part of the development of electronics products is the design of injection molded thermoplastic parts ranging from chassis panels and enclosures (lo connectors. While much effort has been spent on analyzing the end-of-life issues associated with thermoplastic parts, such as disassembly, separation, and recycling, a primary driver for design-for-environment of plastic parts is the energy expended in the forming process. as well as process parameters selection on the process energy efficiency through a framework that integrates a 3-d solid modeling environment, numerical analysis of the filling and poat-filling behavior, and an energy-based process model. The process model that is presented describes the modes of energy utilization in injection molding processes as they relate to design-for-environment concerns. Case studies are performed on a center-gated plastic disk, and a small plastic enclosure to illustrate the use of this framework (10)

2.11 Determination of process parameters based on cavity pressure characteristics to enhance quality uniformity in injection molding

Jian-Yu Chena Ping-Han Hung b Ming-Shyan Huangb (2021)

An appropriate choice of the process parameters is essential to ensure the precision and uniformity of the molded parts. The flow length-to-wall thickness (l/t) ratio for most molded components lies in the range of 100 200. For higher values of the l/t ratio, the flow resistance increases dramatically at the end of the molded part and the injection pressure propagates less easily through the mold cavity. Consequently, the geometric dimensions of the molded parts in the near-gate region are frequently different from those in the far-from-gate region as well as resulting in lower uniformity molded components with a large l/t ratio, the v/p switchover point should be determined based on the quality requirement for the part in the far-from-gate region in order to avoid insufficient (11)

2.12 Enhancing the quality stability of injection molded parts by adjusting switchover point and holding pressure

Jian-Yu Chena Jia-Xiang Zhuang b Ming-Shyan Huangb (2021)

Injection molding process is widely used for the mass production of plastic parts. However, as product designs become lighter, thinner and smaller, and the allowable tolerances shrink, the need to improve the quality consistency of the injection molding process has emerged as a critical concern. In mass production, injection molding is typically performed using fixed settings of the machine inputs. However, in practice, the properties of the plastic melt invariably change from batch to batch and even from shot to shot. Among these four parameters, the v/p switchover point and holding pressure induce a particularly rapid dynamic response, and thus provide an effective means of compensating for process deviations from shot to shot. By changing the melt temperature to simulate environmental noise, it is shown that the two parameter settings enable the consistency of the part weight to be restored to a range of 0.01 g within six shots. Moreover, the yield rate is enhanced from 60% to 90%. (12)

2.13 Injection moulding simulation results as an input to the injection moulding process

B Nardina K Kuzmanab Z Kampusb (2002)

In the presented research work the authors tried to develop the software which will suit all the needs of the injection moulding when optimizing the part-mould-technology system. The simulation results consist of geometrical and technological data. Geometrical data are useful for both: the part as well as the mould designers, because technological data help the moulders to understand the process parameters. The paper provides evidence that the program with its open structure suit the needs of the laboratory environment as well as of the real production. (13)

2.14 Optimization of clamping force for low-viscosity polymer injection molding

Jian-Yu Chena Kai-Jie Yang b Ming-Shyan Huangb (2020)

Injection molding provides a low-cost and efficient technique for the mass production of plastic parts. In general, appropriate processing condition determines the quality of molded parts. Especially, the clamping force is essential in guaranteeing the appearance and geometric accuracy of the molded components. This problem is particularly apparent for low-viscosity resins, which have a high flow capability and thus readily form flash defects even under very small separations of the mold halves during the molding process. Accordingly, this study has proposed a robust method for determining the optimal clamping force setting for the injection molding of low-viscosity polymer resins such as PP and TPU. (14)

2.15 Shear viscosity measurements of polymer melts using injection molding machine with adjustable slit die

Johanna Aho SeppoSyrjala (2011)

Viscosity measurements for two polystyrenes and two polypropylenes were made using a tailor-made slit die with adjustable height of 0.75, 1.0 and 1.5 mm. Viscosity values calculated from the pressure recordings with all three dies showed satisfactory superposition, which verifies the initial assumption of no slip at the wall. For all the materials, the results were also very well comparable with the ones obtained by capillary rheometer and rotational rheometer. This study showed that a relatively simple, tailor-made slit die can offer an attractive alternative as a rheological tool for small polymer processing companies that cannot invest in proper stand-alone rheometers. Detachable slit inserts enable easy cleaning, thus eliminating one of the major downsides usually associated with slit dies. The shear rate and pressure range in this study were limited by the capacity of the molding machine used: In order to obtain higher shear rates and to measure viscosity under elevated pressure, a machine with larger shot volume and injection pressure capacity is needed. (15)

2.16 Simulation of the gas-assisted injection molding process using a mid-plane model of a contained-channel part

Marcilla A.Odjo-Omoniyi R.Ruiz-Femenia J.C.García-Quesada (2006)

Computer-aided engineering (CAE) simulation and experimental studies have been carried out on the cavity filling and gas packing steps in the gas-assisted injection molding of a contained-channel part. A mid-plane model of the three-dimensional geometry of the mold cavity has been proposed to be analyzed by the finite element method. The shot size, the distribution of the gas bubble and the residual wall thickness were calculated using commercial simulation software (Mold flow Plastics Insight Version 4.1).

The outcomes predicted by simulation were compared with the experimental results indicating the good predictive capability of the proposed model. (16)

2.17 An economic analysis comparing the cost feasibility of replacing injection molding processes with emerging additive manufacturing techniques

Matthew Franchetti & Connor Kress (2016)

Additive manufacturing (AM) has proliferated in recent years and is displacing traditional manufacturing methods in numerous applications due to improvements in process efficiencies and cost reductions related to the evolving AM processes. This study explores the cost structure and break-even points of AM versus traditional methods. The comparative analysis examined the cost requirements of AM versus injection molding to manufacture various lot sizes of parts. Break-even points based on lot sizes and the relationship to the overall cost structure were also calculated. This research concludes that break-even points may be calculated based on part mass, density, and lot size. (17)

2.18 Setup reduction in injection molding machine type jt220rad by applying single minutes exchange of die (SMED)

Uly Amrina1, Didi Junaedi1 and Eko Prasetyol (2018)

Injection Molding Machine Type JT220RAD is one of the machines used in the production process of fuel pump module variant 860L Gasoline, 800L Gasoline and D87A Set Plate in PT ANI. The machine is having problems with too much setup time in 99.93 minutes per variant turnover. Therefore, this study was made to reduce the setup time of injection molding machine type JT220RAD by 35%. The researchers applied the

SMED (Single Minute Exchange of Die) method with 5 steps, identifying setup activities and measuring setup time, separating internal and external setup activities, converting internal setup to external, performing kaizen with streamlining setup process, evaluating kaizen done. The result of this SMED implementation is decrease setup time 37.66% (38 minutes per variant turnover) and increase productivity 3.17%. (18)

2.19 Design of pin-point gate injection mold for shells of earplugs

Xian Feng, Min Yang & Min Zou (2018)

The whole design scheme of injection mold for shells of earplugs was determined by analyzing its structure and injection molding technology. In order to adapt to the high product batch and high requirement for appearance quality, an angle ejector of injection mold side core pulling was designed. The layout of a type of pin-point mold with one module and four cavities was determined. The selection of parting surface, the calculation of the cavity number and layout, mold core and cavity design and the design of cooling system were mainly introduced. This paper introduces the structure and working principle of the injection mold for the earplug shell in details. The automatic remolding of the pouring material is realized. The mold structure is compact which reduces the cost of die manufacturing. It has been proved by practice that the mold is reasonable in structure and the quality of plastic part is stable and has good economic benefits (19).

2.20 Effect of process parameters on cavity pressure in injection molding

Q Wang, M Zhen, Z Wu, Y Cai -AIP Conference Proceedings (2017)

In this study, an experimental work is performed on the effect of injection molding parameters on the polymer pressure inside the mold cavity. Different process parameters of the injection molding are considered during the experimental work (packing pressure, packing time, injection pressure, mold temperature, and melt temperature). A set analyses are carried out by combining the process parameters based on the L16 (45) Taguchi orthogonal design. The cavity pressure is measured with time by using Kistler pressure sensor at different injection molding cycles. The results show the packing pressure is significant factor of affecting the maximum of diverse spline cavity pressure. The results obtained specify well the developing of the cavity pressure inside the mold cavity during the injection molding cycles. (20)

2.21 Improved contact lens injection molding production by 3D printed conformal cooling channels

Yf Lin, Jr Wu, Bh Liu, Wcj Wei (2017)

In this research, the combination of finite element simulation and 3D printing enabled casting was adapted to overcome the restrictions on traditional machining processes, we presented a fabrication method for complex injection molding mold pieces with a shorter time and lower processing cost than traditional machining and laseror e-beam-based 3D printing methods. Based on the plastic injection molding simulation results, which considered the cooling rate of the contact lens mold and the cooling time of each part of the mold, we concluded that the design of conformal cooling channel locating at contact lens mold had little influence on the period of injection. However, the simulation results showed that the cooling efficiency did not increased significantly when a conformal cooling channel was placed in the sprue bush. But the cooling time reduce about 20%. In other words, the cooling of sprue bush dominated production cycle time. Therefore, a smart mold with real-time temperature monitoring was made with 3DP and mold flow simulation for this work. (21)

2.22 A deformation compensation method for wax pattern die of turbine blade

R Jiang, D Zhang, K Bu, W Wang, J Tian (2017)

The turbine blade is one of the most important parts in turbine machinery. Casting deformation is an important index to evaluate the quality of the turbine blade. In order to control the deformation of the turbine blade during investment casting, a novel compensation method based on reverse deformation was proposed in this study. Firstly, the process of deformation compensation was discussed. To overcome the disadvantage of the iteration compensation process, a one-step compensation model was developed based on Taylor expansion. Moreover, a smooth deformation function for the compensation model was regressed based on casting simulation results. Finally, two blades were optimized according to the proposed methodology.

The results indicated that the casting deformation was reduced significantly. Hence, a more accurate blade can be achieved. (22)

2.23 Optimization of die casting process in zamak alloys

Yekta Berk Suslu, Mehmet Sirac Acar, Mithat Senol, Muammer Mutlu & Ozgul Keles (2018)

The casting industry is one of the major industries in the world with a great impact in everybody's life. Die casting is a process where a permanent mold is used, and melted metal is injected by pressure, allowing smaller cycles and continuum parts production. This study is focused in die casting applied to automobile industry where many casted parts are used in their components. The study was developed in order to maximize the quality of small parts injected in Zamak alloy for automotive components. Using simulation, the runners' location was improved as well as gas relief. (23)

2.24 Design the control system circuit of vacuum type injection machine

CZ Wu, SS Lu, FY Zhang (2017)

Design the control system of EWVCM-600 vacuum casting machine according to the principle of vacuum injection. EWVCM-600 vacuum-injection machine control system is mainly consisted of the mechanical structure and circuit structures.Due to the systems for the process is relatively fixed, and the control operation is relatively simple, thus the control part applies the simple and stability relay to control start or stop of the vacuum pump and start or stop of gas equipment. AC variable speed motor drives the agitator, the stirring speed is controlled by operating the governor on the panel. Analyze and test the working principle and process of EWVCM-600 vacuuminjection machine, and design the mechanical structure and circuit diagrams. It achieves the desired objectives. (24)

2.25 An automated manufacturing analysis of plastic parts using faceted surfaces

JM Mercado-Colmenero, JAM Muriana (2017)

In this paper a new methodology of automated remold ability analysis for parts manufactured via plastic injection molding is presented. Remold ability analysis is based on a sequential model to catalog nodes and facets of the given mesh. This methodology uses a discrete model of plastic part, which provides an additional advantage since the algorithm works independent of the modelling software and creates a new virtual geometry providing information on its manufacture, exactly like CAE software. All elements of the mesh (nodes and facets) are stored in arrays, according with their remold ability category, with information about their manufacture for possible uses in other CAD/CAE applications related to design, machining and costs analysis of injection molds. (25)

2.26 Design of Three Cavity Die casting Die for Rotors

ST Varghese, B Singh (2017)

Die casting components play an important role in every aspect of modern world. Their influence ranges from house hold utensils to automobile components. Requirement of today's world is production, accuracy and interchangeability, which helps to meet the competition. In order to meet these challenges, die casting process plays an important role in production. This paper gives an engineering approach towards design of die casting die for rotors and deals with the design of a three cavity die casting die by using the CATIA V5 Software. In this work the computer aided design of three cavity die casting die is the replacement of traditional design of two cavity die casting die for rotors which is existed .In this paper we also replace cavity and core impressions on block with inserts. (26)

2.27 Mold filling simulation of low-pressure injection molding (LPIM) of alumina: Effect of temperature and pressure

M Sardarian, O Mirzaee, A Habibolahzadeh (2017)

Filling stage is the most important stage in powder injection molding (PIM) process so that many defects may occur in this stage. In the present study, filling stage for low pressure injection molding (LPIM) of alumina was simulated by finite element method (FEM). Experimental tests were conducted on alumina feedstock with 60 vol% powder.The melt front shape obtained from the simulation matched the experiments well in top view but was slightly different in side view. Comparison between simulation and experimental results showed good agreement with regard to the melt front locations. Simulation has been found to correctly describe trends such as an increase in the pressure required to fill molds as temperature decrease. (27)

2.28 Common defects in metal injection molding (MIM)

KS Hwang (2012)

Inconsistent product quality, including poor dimension control, distortion, and internal and external defects, has tended to be underestimated by metal injection molding (MIM) practitioners. These defects may originate in the early processing steps, but they often do not manifest until after rebinding or sintering. Thus, the solutions are difficult to provide. This chapter presents an overview of MIM defects. Explanations for these defects are described and, where available, remedies and suggestions are provided. (28)

2.29 Design and fabrication of a low-volume, high-temperature injection mould leveraging a 'rapid tooling' approach.

H Kalami, RJ Urbanic (2019)

The costs for low-volume production moulds (1–200 production components) are related to the mould material, the process planning time and the fabrication costs. A general solution is provided, with a case study focusing on an over moulding process in which the injection material being moulded is Techno melt-PA 7846 black. This pattern is used to form a resin-based insert which is to be assembled into a mould base frame. Customized inserts can be readily built and exchanged to provide a rapid response to a customer request. An assessment of the digital model, the manufacturing, assembly and the final validated assembly model is provided. (29)

2.30 The design of conformal cooling channels in injection molding tooling.

X Xu, E Sachs, S Allen (2001)

Solid Freeform Fabrication technologies have demonstrated the potential to produce tooling with cooling channels, which are conformal to the molding cavity. 3D Printed tools with conformal cooling channels have demonstrated simultaneous improvements in production rate and part quality as compared with conventional production tools. Conformal cooling lines of high performance and high complexity can be created, thus presenting a challenge to the tooling designer. A systematic, modular approach to the design of conformal cooling channels is presented. Cooling is local to the surface of the tool, so the tool is divided into geometric regions and a channel system is designed for each region. Each channel system is itself modeled as composed of cooling elements, typically the region spanned by two channels. The methodology is demonstrated through application to a complex core and cavity for injection molding. (30)

2.31 Design of Precision Injection Mould for Hot Runner of Automobile Back Door Inner Plate based on Front Die Core-Pulling

Huangying Wen (2020)

Taking the inner panel of the Automobile Back Door as the research object, the design flow, design characteristics and key technology of the injection mold were analyzed, and a set of hot runner injection mold for the inner panel of the automobile back door was designed. Based on the analysis of die structure, forming parts, pouring system and side core pulling, the key and difficult solution of die was determined. The hydraulic cylinder +Slide Block core-pulling mechanism, inclined guide post +slide block and inclined push rod remolding mechanism were used to solve the remolding problem with many reverse buckles in different internal directions successfully, the position of Weld Line is adjusted successfully, the weld line is reduced, and the quality of plastic parts is guaranteed. (31)

2.32 Design Principles of a-three-plate cold runner mold

DA Nguyen, QA Nguyen (2020)

Due to the high consumption by humans for plastic hangers, the study on equipment and process, especially the injection molding machine, to manufacture these products has been attracting the researcher's attention. To enhance the performance of the machine, some parts of the injection molding should be scrutinized. Therefore, the thesis aims to size the three-plate cold runner molding machine to serve the production of the plastic hanger. To design the practical runner, the theory-based knowledge must be clarified. Therefore, the prioritized aim of the thesis is to calculate the research of theory, especially the theoretical parameters of the runner based on the fundamental background of heat transfer and mass transfer (35)

2.33 Design and Fabrication of Injection Molding Tool for Cam Bush with Baffle Cooling Channel and Submarine Gate

S Selvaraj, P Venkataramaiah - Procedia Engineering (2013)

This research works is focused on design and fabrication of an automatic injection moulding tool for production of CAM BUSH which is used in electrical engines as a connector. An injection mould is a tool which is used for production of plastic components in large numbers in a short span of time. The material, Oil Hardened Nonshrinking Steel (OHNS) for the core and cavity, EN353 for the guide pillar, guide bushes, core pins, ejector pins, locator ring and the sprue bush and Mild Steel for other plates are selected. The elements of injection moulding tool have been designed, fabricated and assembled. Submarine gate and Baffle circular whole cooling system has been provided in the moulding tool to increase the productivity and good surface finish. The required Nylon-66 Cam bush component is produced with this moulding tool by properly controlling the various parameters of injection moulding machine. (33)

2.34 Design and Fabrication of Plastic Injection Molding Tool for Pump Gaskets

H Kummara, D Gowd (2014)

This Project gives the information about to design and fabrication of Injection Molding tool or die for Production of pump gaskets by using Plastic material. For Increasing the Production rate, designing and manufacturing the multi cavity die and also Using PP (polypropylene) to overcome the existing plastic material drawbacks. The injection molding tool/die contains the core plate, cavity plate, top and bottom supported plates, channels or runners, sprue, vents, ejector and its pins and horn pins, etc. The gaskets are used to prevent leakages by provide a tight fitting joint between two surfaces. The Greek word of 'Plastic' meaning is 'able to be shaped and molded'; in so many different plastic materials .The selected two bolt oval gasket component is making with injection molding process by controlling parameters of plastic injection molding machine. (36)

2.35 Optimization of Gate, Runner and Sprue in Two-Plate Family Plastic Injection Mould

MA Amran, M Hadzley, S Amri, R Izamsha (2010)

This paper describes the optimization size of gate, runner and sprue in two-plate family plastic injection mould. An Electronic Cash Register (ECR) plastic product was used in this study, which there are three components in electronic cast register plastic product consist of top casing, bottom casing and paper holder, Rhinoceros software as post processing tool was used to design gate, runner and sprue and Moldex simulation tool was used to analyze the plastic flow. As result, some modifications were made on size of feeding system and location of cavity to eliminate the short- shot, over filling and welding line problems in two-plate family plastic injection mould. (35)

2.36 Literature gap

Title	Author	Year	Structure	Material	Test
Practical Applications of Taguchi Method for Optimization Processing Parameters for Plastic Injection Molding: A Retrospective Review	NgChina Fei,Nik Mizamzul Mehat, Shahrul	2013	Injection molding die	MS material	Processing parameters for injection moulding.

Investigating the cutting phenomena in free-form milling using a ball-end cutting tool for die and mold manufacturing	Adriano Fagali de Souza Anselmo Eduardo Diniz, Alessandro Roger Rodrigues & ReginaldoTeixeiraCoelho	2007	Die Manufacturing	MS material	A ball-end cutting tool for manufacturing of dies and molds
Productivity enhancement in dies and molds manufacturing by the use Brenier of C1 continuous tool path	M Boujelbene, A Moisan, N Tounsi, B	2004	Die Manufacturing	Ms material	To Improve the surface roughness the process of milling
Modern machining of die and mold tools.J.Mater ProcessTechnol structures with shape memory effects	Krajanik P, Kopac J	2004	Die Manufacturing	MS material	Enhance ment of processes of die manufacturing by non-conventional machining
Optimization of Die Casting Process in Zamak F. J.G.Alloys	Pinto, Helder Silva	2017	Die casting	Alloys	Casting of die alloy

Design the control Vacuum type injection machine Zhang, Fu You	Wu,ChangZhong	2017	Die manufacturing	MS	Tensile and shear loading
Design of Three Cavity Die-casting Die for Rotors	(S. Chandra Sekhar Sk.Surjan,	2017	Die manufacturing	MS	Die-casting Die for Rotors
The design of conformal injection molding injection molding tooling.	Zheng, qing	2001	Die components	Ms	Cooling channels

2.37 Reference model

Injection molds, which are usually made from steel, contain cavities that will form the parts. Melted plastic is injected into the mold, filling the cavities. The mold is cooled, and the parts are ejected by pins. This process is similar to a Jello mold which is filled then cooled to create the final product.

2.38 Design, Analysis and Fabrication of Cam Dies in Injection

Figure2.1Reference Model

Chapter 3

DESIGN AND FABRICATION

3.1 List of Components

- Ejector pin
- Ejector retaining plate.
- Ejector plate
- Mold core
- Ejector box
- Support plate
- Guiding pins
- Locating ring
- Springs
- Clamping plates (fixed plate)
- Cooling channels

3.2 Components Detail

3.2.1 Ejector Pin

Ejector pins are vital in creating parts. They are an integral component of the ejection system in mold, which determines the final outcome of products in an injection molding process. Injection molding is a manufacturing process that involves injecting molten plastic in a metal mold to assume the shape of the mold.

3.2.2 Ejector Pin 3D Model

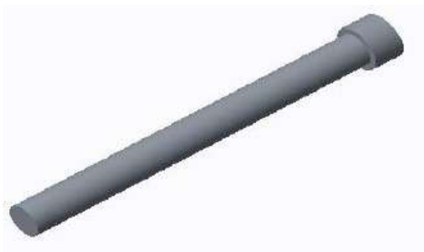


Figure 3.1: Ejector Pin 3D Model

3.2.3 Ejector Retaining Plate

An ejector retainer plate holds the heads of ejector pins in place. The main function of this plate is to lock the heads of the pins so they don't come out during the injection molding process. The ejector bar plate has almost the same role as an ejector retainer plate

3.2.4 Ejector Retaining Plate 3D Model

Figure 3.2: Ejector Retaining Plate 3D Model

3.2.5 Ejector Plate

In an ejection system in mold, ejector plates function alongside the ejector pins. It holds the head of the pins to prevent them from coming out during the ejector pins injection molding process.

3.2.6 Ejector Plate 3D Model

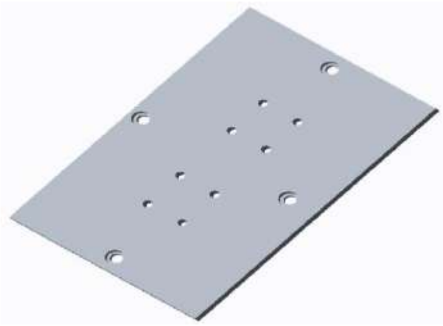


Figure 3.3: Ejector Plate 3D Model

3.2.7 Mold Core

The Mold Core & Cavity are the shaped sections in either half of the mold tool which give the plastic product its final shape. The hot molten material is injected into the core & cavity and then sets hard into shape. The design of the core & cavity is essential in the correct formation of the product.

3.2.8 Mold Core 3D Model

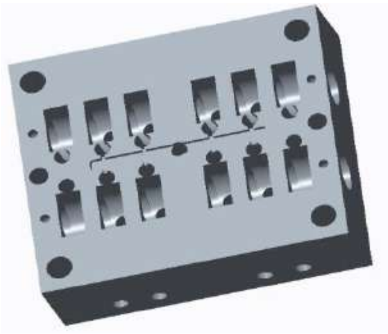


Figure 3.4: Mold core 3D Model

3.2.9 Support Plate

The backup plate or support plate is used to support the cavity plate, attach the hole for the return pin's spring, and cooling channel when in cavity plate cannot make it.

3.2.10 Support Plate 3D Model

Figure 3.5: Support Plate 3D Model

3.2.11 Guiding pins

Guide pins are cylindrical rods used in an assembly to align components. The mechanical fastener may help ensure stability, limit contact damage, and maintain precise positioning.

3.2.12 Guiding Pin 3D Model

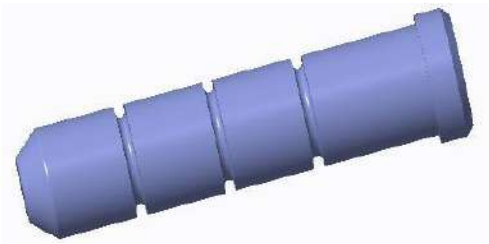


Figure 3.6: Guiding Pin 3D Model

3.2.13 Locating Ring

A locating ring is a circular member fitted on to the front face of the mold over the sprue bush. Its purpose is to register (or locate) the mold in the correct position on the injection machine, to ensure proper alignment between the nozzle and the sprue bush, thereby eliminating leakage.

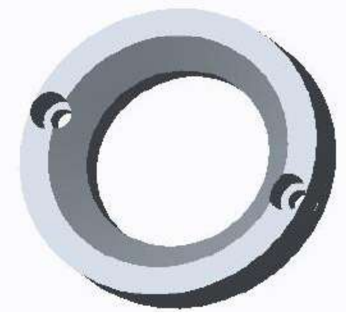


Figure 3.7: Locating Ring 3D Model

3.2.14 Spacer Block

Spacer Block is mounted between the movable clamping plate and the movable cavity plate to give space and allow the ejector plate to move when ejecting the part.

3.2.15 Spacer Block 3D Model

Figure 3.8: Spacer Block 3D Model

3.2.16 Springs

A spring is a device consisting of an elastic but largely rigid material (typically metal) bent or molded into a form (especially a coil) that can return into shape after being compressed or extended. The most common use of springs in an injection mold is to retract the ejector plates.

3.2.17 Springs 3D Model

Figure 3.9: Springs 3D Model

3.3 Moving Unit Assembly CAD Model

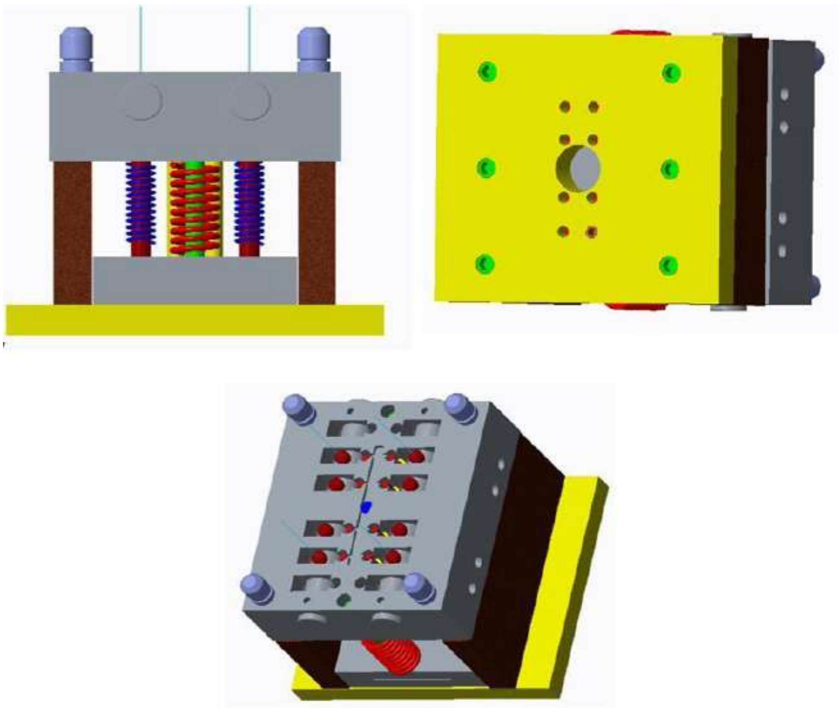


Figure 3.10: Moving Unit Assembly CAD Model

3.4 Fixed Plate Unit

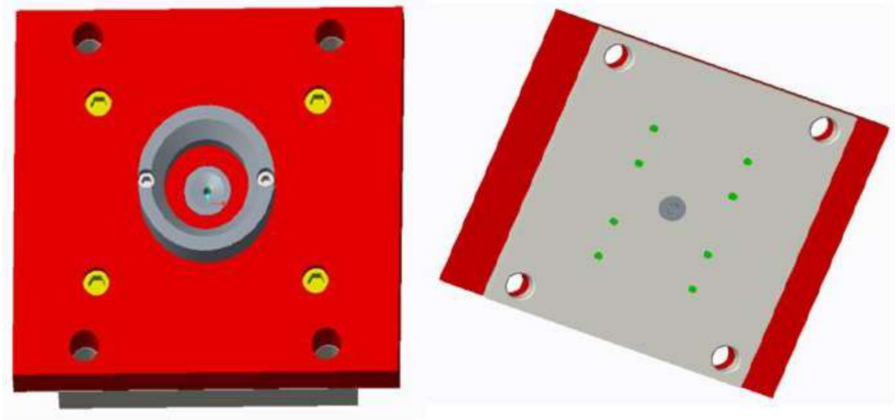


Figure3.1Fixed Plate Unit

3.5 Final Assembly Cad Model

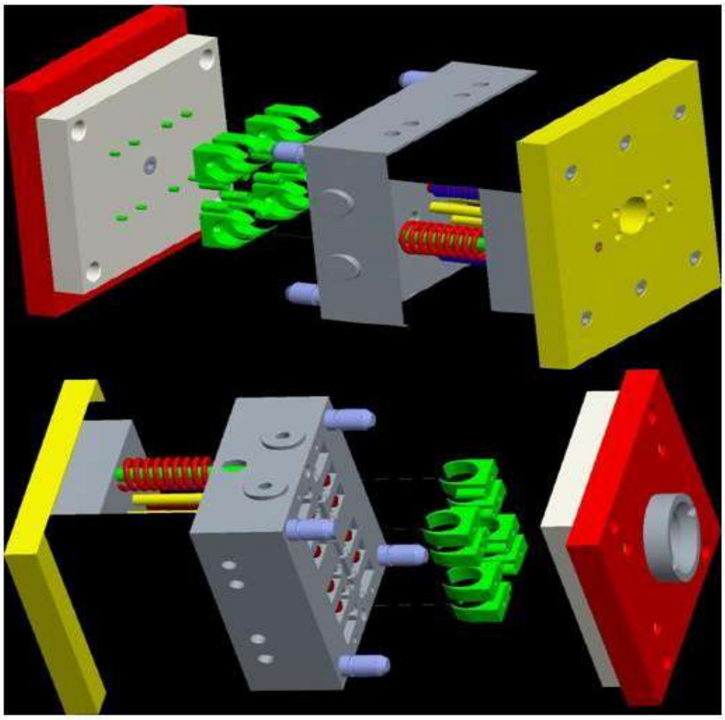


Figure 3.12: Final Assembly CAD Model

3.6 Cost Analysis

3.6.1 Material

The type of material used to make the die have a significant impact on the cost.

3.6.2 Die size

The larger the item or part to be molded, the larger its mold will be to accommodate the part. Larger parts tend to require more injected materials to complete production cycles. Larger mold designs usually come at an increased cost compared to fabricating the same design but with smaller dimensions.

3.6.3 Machining process

CNC machines are the most commonly used tools for manufacturing aluminum and stainless-steel molds with high precision levels. CNC machining removes material by a spinning tool and fixed part. Machining can produce molds where the cavity design is highly complex, but they might require multiple tool changes that can slow down the process, which means that costs increase in line with complexity.

3.6.4 Tooling and equipment

The cost of tooling and equipment needed to manufacture the die, such as CNC machines, cutting tools, and inspection equipment, will also be factored into the overall cost.

3.6.5 Labor costs

The cost of skilled labor required to manufacture the die, including design, programming, machining, and inspection, will be a significant contributor to the cost.

Chapter 4

EXPERIMENTATION & RESULTS

4.1 Experimental Methodology

The experimental methodology for an injection molding machine mold involves various steps to optimize the mold design and ensure the production of high-quality plastic parts. Here’s a general outline of the experimental methodology for the mold.

4.1.1 Objective Definition

Clearly define the objectives of the mold experiment. Identify the specific parameters you want to optimize, such as part quality, dimensional accuracy, cooling efficiency, mold filling behavior, etc.

4.1.2 Material Selection

Choose the appropriate mold material based on the requirements of the injection molding process and the properties of the plastic material being used.

4.1.3 Mold Design and Fabrication

Design the mold based on the specifications of the plastic part you intend to produce. The mold should consider factors such as gate design, runner system, cooling channels, ejection mechanism, and venting. We selected the high carbon steel 1095 for the mold manufacturing.

Typical composition of high-carbon steel - 1095

Material	Iron	Carbon	Manganese	Sulfur	Phosphorus
1095 steel	98.4%	0.95%	0.5%	<0.05%	<0.04%

4.1.4 Prototype Mold

Depending on the complexity of the mold design, it is often useful to create a prototype mold using less expensive materials to validate the design and make any necessary adjustments before final production.

4.1.5 Mold Setup

Set up the mold on the injection molding machine and ensure it is properly aligned and secured.

4.1.6 Baseline Experiment

Run a baseline experiment with default mold settings to obtain initial data and assess the initial part quality and mold performance.

4.1.7 Experimental Runs

Conduct a series of experiments by varying mold parameters such as gate size, cooling channel design, venting, ejection timing, etc. Record all relevant data during each run, such as injection pressure, temperature, cycle time, part quality, and any observations about mold behavior.

4.1.8 Data Collection and Analysis

Collect and compile the data from all experimental runs. Perform statistical analysis and identify trends or relationships between mold parameters and part quality.

4.1.9 Optimization

Based on the data analysis, determine the optimal mold design and setup for achieving the desired part quality and production efficiency.

4.1.10 Validation Runs

Conduct validation runs using the optimized mold parameters to ensure consistency and repeatability of results.

4.1.11 Quality Testing

Perform quality testing on the molded parts to validate that they meet the required specifications and standards.

4.1.12 Documentation

Document the entire experimental methodology, including the mold design, experimental setup, data collected, analysis, and the final optimized mold parameters.

4.1.13 Continuous Improvement

Injection molding molds can be continuously improved based on feedback from production and any new requirements. Monitor the mold performance regularly and make necessary adjustments or modifications.

4.1.14 Safety Considerations

Throughout the experimental process, prioritize safety by adhering to mold setup and handling guidelines and using appropriate personal protective equipment (PPE).

4.2 Assembly Manual of Project

- Prepare the mold component.
- Identify the correct positioning.
- Install the mold base.

- Mount core and cavity inserts.
- Install cooling system.
- Mount ejector pins.
- Check for proper clearances.
- Lubricate moving parts.
- Final inspection.
- Close the mold.

4.3 Experimental Manual

4.3.1 Material Selection

Determine the material you will be using for injection molding. Different materials have different melt temperatures, flow properties, and shrinkage rates, which can influence the molding process and the quality of the final product.

4.3.2 Machine Setup

Set up the injection molding machine according to the specifications provided by the manufacturer. Ensure that the machine is clean and calibrated properly. This includes adjusting the temperature settings, pressure, injection speed, cooling time, and any other relevant parameters based on your experiment objectives.

4.3.3 Mold Preparation

Install the mold onto the injection molding machine. Make sure the mold is securely fastened and aligned correctly. Check that all the components, such as the sprue, runner system, and cavity, are clean and free from any debris or contamination.

4.3.4 Process Parameters

Determine the process parameters you want to investigate. These can include injection pressure, injection speed, cooling time, melt temperature, and any other variables that may impact the molding process and product quality. It's recommended to create a plan to systematically vary these parameters to collect meaningful data.

4.3.5 Data Collection

Set up a data collection system to capture relevant process parameters during the molding cycle. This may involve using sensors, data loggers, or software provided by the injection molding machine manufacturer. Record important variables such as temperature, pressure, cycle time, and any other relevant measurements.

4.3.6 Experiment Execution

Run the injection molding process according to the predetermined parameters and collect data for each experimental condition. Repeat the process multiple times to account for variability and ensure statistical significance.

4.3.7 Quality Control

Inspect the molded parts to evaluate their quality. This can involve visual inspection, dimensional measurements, mechanical testing, or any other relevant quality criteria. Document the results for each experimental condition.

4.3.8 Data Analysis

Analyze the collected data to draw meaningful conclusions. Identify trends, correlations, and relationships between the process parameters and the quality of the molded parts. Statistical methods such as analysis of variance (ANOVA) or design of experiments (DOE) can be used to analyze the data.

4.3.9 Conclusion and Recommendations

Based on the results of your experiment, draw conclusions, and make recommendations. If needed, suggest modifications to the process parameters, mold design, or material selection to improve the overall performance.

4.4 Experimental Parameters

- Temperature.
- Injection Speed.
- Injection Pressure.
- Holding Pressure and Time.
- Cooling Time.
- Mold Venting.
- Mold Surface Finish.
- Material

4.5 Experimental Calculations

4.5.1 Table of experimental results

4.5.2 Temperature Calculation 6

Table 4.1: Temperature Calculation

MATERIAL MELTING TEMPREEATURE IN °C					EJECTION TEMPREATURE °C
	SEGMENT 1	SEGMENT 2	SEGMENT 3	SEGMENT 4	
MEAN TEMPRATURE	227	227	179	0	50
SET TEMPREATURE	230	230	200	0	50
TOLERENCE IN TEMPRATURE	±30	±30	±30	0	

Fig 4.1. Temperature Calculation

4.5.3 Dais Setting

Table 4.2: Dias Setting

NOZZEL ADD PRESSURE	NOZZEL REVERSE PRESSURE	FLUX-RATE (PERCENTAGE)
60	70	50%

Fig.4.2. Dias Setting

4.5.4 Melt/Take/Cool Calculation of Lead Screw Rod

Table 4. 3: Melt /Take/Cool Calculation

	F. TAKEN	MEL 1	MEL 2	B. TAKEN
PRESSURE(BAR)	0	55	55	55
FLUX (%)	0	50	50	55
PLACEMENT (mm)	0.0	50.0	160.0	0.0
B. PRESSURE (BAR)	ENABLE	20	20	

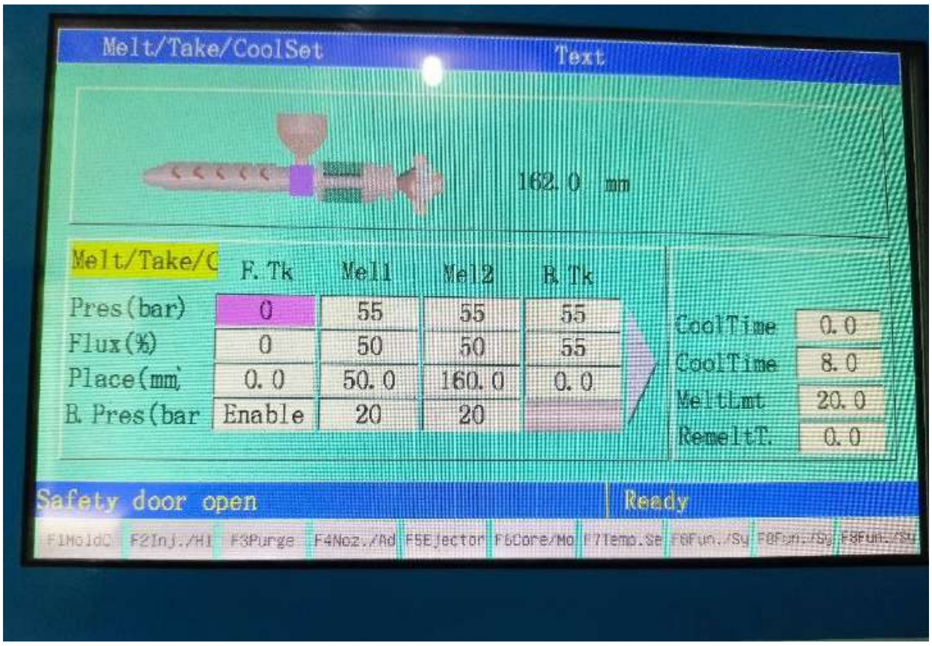


Fig.4.3. Melt /Take/Cool Calculation

4.5.5 Injection Segments Calculation

Table 4: Injection Segments Calculation

	SEGMENT 6	SEGMENT 5	SEGMENT 4	SEGMENT 3	SEGMENT 2	SSEGMENT 1
PRESSURE (BAR)	0	0	0	0	65	65
FLUX (%)	0	0	0	0	50	45
PLACEMENT (mm)	0	0	0	0	85	75
TIME (sec)	0	0	0	0	0	0

Fig.4.4. Injection Segments Calculation.

4.5.6 Mold Close Calculations

Table 5: Mold Close Calculation

	SLOW	QUICK	LOW PRESSURE	HIGH PRESSURE
PRESSURE (BAR)	50	50	80	99
FLUX (%)	30	30	30	45
PLACEMENT (mm)	150	160	30	
LIMT TIME (sec)		30		
	DI	FF LOCK: DISABLE		

4.6 Output product

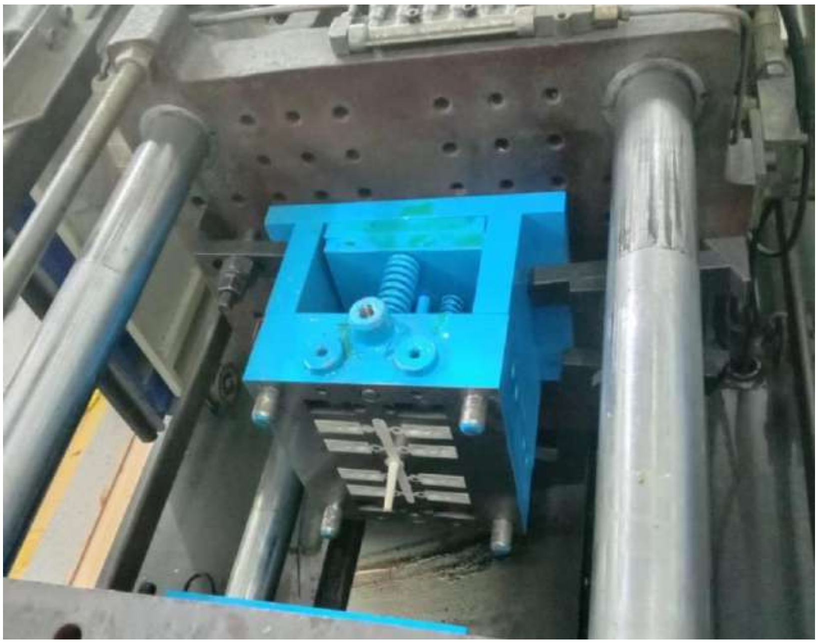


Fig.4.5. Output Product 1

Fig 4.6. Output Product 2

Fig.4.6. Output Product 3

4.7 Reference Model

Design, Analysis and Fabrication of Safe Holder and Cam Dies in Injection Molding

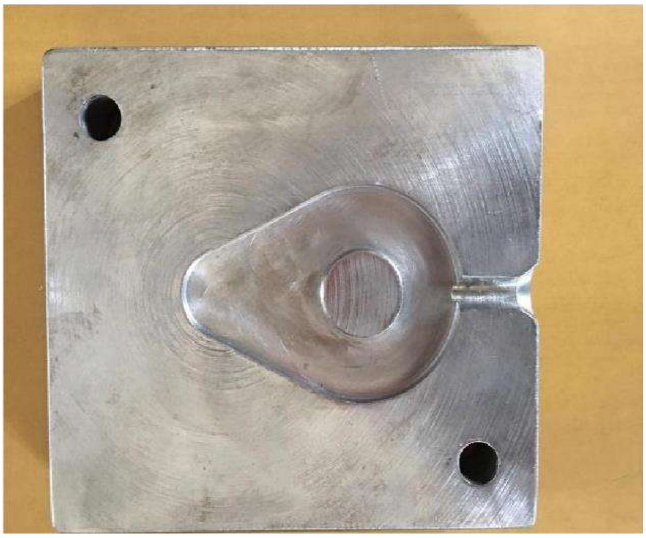


Figure 4.7 Cam Cavity In Die Upper Half With Ejector Pin

Figure 4.8 Cam Cavity Lower Side

• Comparison of analysis results with physical results

Table 6: Comparison of analysis results with physical results

Parameter	Analysis Results	Physical Results
DIMENSIONS OF DIES (MM):	1.70X120X4 2.70X90X4	1.70X120X4 2. 70X90X4
Initial temperature (℃)	260	200
Cycle time (sec)	80	52.5
Injection pressure (Bar)	150	90-110

4.8 Cost Analysis of Output Product

Electricity cost = 3.75rupees per cycle

Material cost = 2.5per cycle

Melting time = 20-25minuutes

Product cost = 6.25 rupees per product Overall production cost = 50 rupees

4.9 Effect of Shrinkage with Respect to Temperature and Pressure

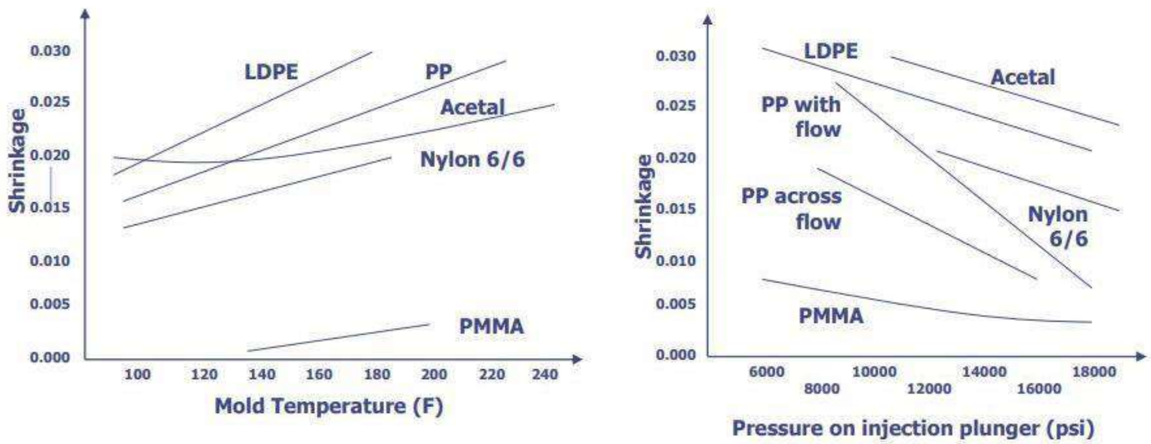
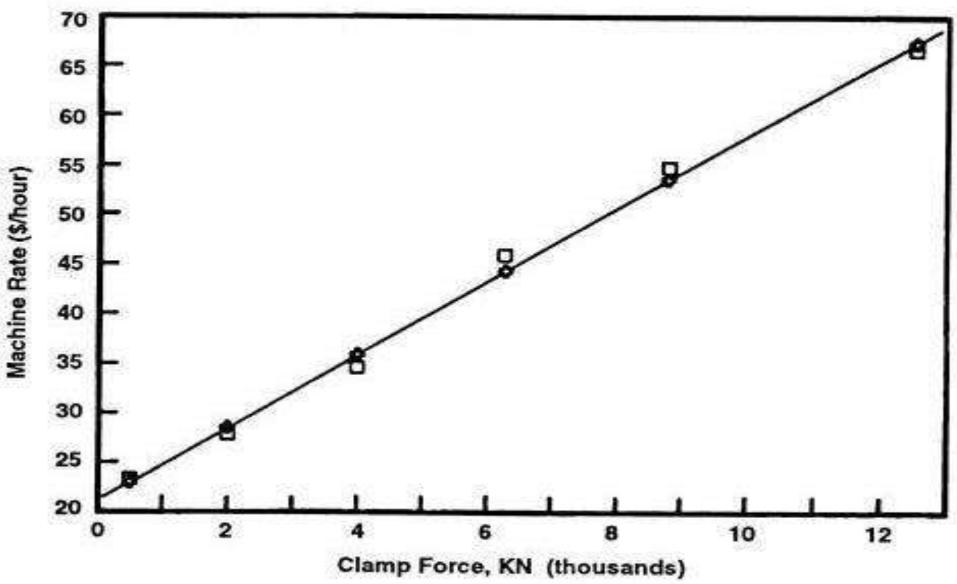


Figure 4.8. Effect of Shrinkage with Respect to Temperature and Pressure

4.10 Clamping Force Cycle



4.11 Clamping Force Cycle

4.12 Typical Temperature / Pressure Cycle

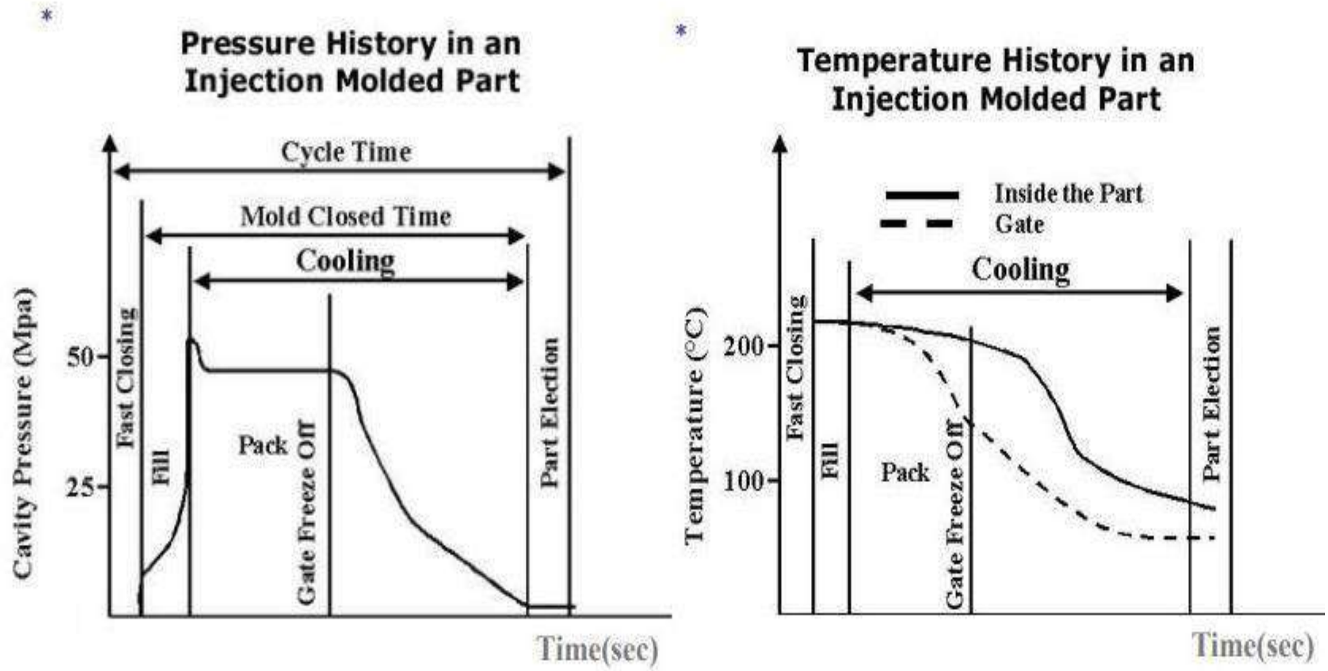


Figure 4.10 Typical Temperature / Pressure Cycle

4.13 Improvement/Efficiency with Reference Model

- Mass production.
- Less costly.
- Less time consuming. • Highly reliability
- Defect less.
- Easy mold maintenance and cleaning.
- Household usage.
- Low production cost.
- Easy material availability.
- scrap material can be used.

4.14 Efficiency of fabricated mold

4.14.1 Cycle Time

The cycle time is the time it takes for a complete injection molding cycle, including filling the mold with molten material, cooling, and ejecting the finished part. The shorter the cycle time, the higher the efficiency. Reducing cycle time can be achieved through optimized mold design, appropriate material selection, and efficient machine operation.

4.14.2 Cavitation

The number of cavities in the mold affects the production output. Higher cavitation molds produce more parts in each cycle, leading to increased efficiency. However, it's important to balance cavitation with factors like machine capacity and part quality.

4.14.3 Part Quality

The mold should be designed to produce high-quality parts consistently. This includes aspects such as dimensional accuracy, surface finish, and structural integrity. An efficient mold ensures minimal defects and rejects, reducing waste and maximizing productivity.

4.14.4 Maintenance and Downtime

Regular maintenance of the mold is crucial for sustained efficiency. Proper cleaning, lubrication, and inspection can prevent issues like sticking, wear, and damage. Additionally, reducing downtime during mold changes or repairs contributes to overall efficiency.

4.14.5 Mold Design

An optimized mold design takes into account factors such as part geometry, material flow, cooling channels, and ejection mechanisms. Efficient mold design minimizes material waste, optimizes cooling for faster cycles, and facilitates easy part removal.

4.14.6 Material Selection

Choosing the right material for the part and its requirements can enhance efficiency. Factors like melt flow properties, shrinkage, and compatibility with the mold influence the molding process. Selecting materials that can be processed efficiently can improve productivity.

4.14.7 Energy Consumption

Injection molding machines consume energy during the heating, injection, and cooling phases. Efficient molds minimize energy requirements by optimizing the cooling process, reducing unnecessary heat loss, and using efficient machine settings.

4.14.8 Discussion of experimental results

The experimental setup and procedures in detail. Include information such as the type of injection machine used, mold design, material selection, process parameters (e.g., injection pressure, temperature), and any relevant measurements or observations taken during the experiment.

This can include quantitative data, such as cycle time, part dimensions, weight, or mechanical properties, as well as qualitative observations, like part quality, surface finish, or defects. Use tables, charts, or graphs to present the data effectively.

Engage in a discussion of the implications and significance of the experimental results. Consider the practical implications for industrial applications, potential challenges or limitations, and how the results align with existing knowledge or literature in the field. Discuss the strengths and weaknesses of the experimental approach and identify areas for future research or improvement.

Summarize the key findings and conclusions drawn from the experimental results. Emphasize the implications and practical significance of the findings. Highlight any recommendations for process optimization, mold design modifications, or further investigations based on the results obtained.

4.15 Justification/ significance of experimental results

4.15.1 Process Optimization

Experimental results provide valuable insights into the performance of the injection machine mold.By analyzing the data obtained from experiments, engineers can identify process parameters that affect the quality, productivity, and efficiency of the molding process. This information can be used to optimize the process variables, such as injection pressure, melt temperature, cooling time, and mold temperature, to achieve the desired outcomes.

4.15.2 Quality Assurance

Injection molding relies on the precise control of process parameters to produce high-quality parts. Experimental results help validate and verify the quality of the molded parts by evaluating parameters such as dimensional accuracy, surface finish, mechanical properties, and consistency. Deviations from the desired specifications can be identified and corrective actions can be taken to improve the mold design or process conditions.

4.15.3 Material Selection

The choice of material for injection molding greatly influences the performance of the final product. Experimental results provide information about how different materials behave during the molding process, such as their flow characteristics, shrinkage, and processing window. This data helps in selecting the most suitable material for a specific mold design and desired part properties.

4.15.4 Mold Design Improvement

Experimental results can be used to evaluate and refine the mold design. By analyzing the performance of different mold configurations, such as gate locations, cooling channel layouts, and ejection mechanisms, engineers can identify design improvements that can optimize the molding process. This can lead to enhanced productivity, reduced cycle time, and improved part quality.

4.15.5 Troubleshooting and Problem Solving

Experimental results are valuable for troubleshooting issues that may arise during the injection molding process. By conducting experiments, engineers can identify the root causes of defects, such as sink marks, warpage, and voids, and develop effective solutions to rectify them. This iterative approach helps in continuously improving the mold design and process conditions.

4.15.6 Cost Optimization

Injection molding is a cost-sensitive process, and experimental results can be utilized to optimize costs. By analyzing the data obtained from experiments, engineers can identify ways to reduce material waste, minimize cycle time, and improve overall efficiency. This can lead to cost savings in terms of raw material usage, energy consumption, and labor.

4.15.7 Knowledge Generation

Experimental results contribute to the collective knowledge base of injection molding. They provide valuable data and insights that can be shared with the scientific and industrial communities, enabling further research and development in the field. This knowledge exchange helps in advancing the understanding and capabilities of injection molding technology.

Chapter 5

CONCLUSION & FUTURE RECOMMENDATION

5.1 Future Recommendations

5.1.1 Industry 4.0 Integration:

Injection molding machines and molds may become more interconnected with Industry 4.0 technologies, enabling real-time data collection, process optimization, and predictive maintenance to enhance productivity and reduce downtime.

5.1.2 Sustainable Molding Solutions:

As sustainability becomes a more prominent concern, there might be a push towards eco-friendly molding processes and biodegradable or recycled plastics, leading to greener plastic products.

5.1.3 Precision and Micro-Molding:

The demand for smaller and more intricate plastic parts may drive the development of precision and micro molding technologies, requiring highly specialized molds and machines.

5.1.4 Multi-Material and Hybrid Molding:

Advancements in multi-material injection molding and hybrid molding techniques may lead to the production of more complex and functional plastic products, requiring molds capable of handling multiple materials.

5.1.5 Additive Manufacturing for Molds:

The adoption of additive manufacturing for mold-making could enable more design flexibility, faster prototyping, and cost-effective production of custom molds.

5.1.6 Smart and Functional Plastic Products:

Future plastic products may incorporate smart functionalities, such as sensors, actuators, and communication capabilities, opening up new applications and markets.

5.1.7 Improved Part Quality and Consistency:

Continuous advancements in mold design, process control, and material technology could result in higher part quality, tighter tolerances, and greater consistency across production runs.

5.1.8 Reduced Waste and Energy Consumption:

Innovations in injection molding processes and mold designs may help reduce material waste and energy consumption, contributing to a more sustainable and efficient manufacturing industry.

5.1.9 Advanced Mold Materials:

Advancements in materials science may lead to the development of new mold materials with improved durability, thermal conductivity, and wear resistance. These materials could result in longer mold lifetimes and better part quality.

5.1.10 Innovative Cooling Solutions:

Further research into conformal cooling, advanced cooling channel designs, and the integration of cooling technologies like micro-fluidics could significantly reduce cycle times and improve part quality.

5.2 Conclusion

5.2.1 Quality Assurance:

A well-designed and precisely engineered injection machine mold is essential for ensuring high-quality plastic parts. Proper mold design, material selection, and fabrication techniques contribute to achieving consistent and accurate part dimensions and surface finishes.

5.2.2 Productivity and Efficiency:

Optimal mold design and efficient cooling systems can significantly reduce cycle times, resulting in increased productivity and cost savings. Mold designs with advanced cooling solutions, such as conformal cooling, can enhance heat dissipation and shorten the overall production time.

5.2.3 Versatility and Complexity:

Modern injection machine molds are increasingly capable of handling complex geometries, multi materials, and over-molding processes. This adaptability opens up opportunities for producing diverse and innovative plastic products.

5.2.4 Sustainability:

Emphasis on sustainable manufacturing practices has led to the development of eco-friendly mold materials and processes. Sustainable molds contribute to reduced waste generation and energy consumption, aligning with environmental goals.

5.2.5 Integration and Automation:

Injection machine molds are becoming more integrated with Industry 4.0 technologies, enabling real time monitoring, data analytics, and predictive maintenance. Increased automation in mold handling and setup enhances operational efficiency and reduces human errors.

5.2.5 Precision and Miniaturization:

Advancements in mold-making technologies support precision molding and micro-molding, catering to the demand for smaller, intricate plastic components used in various industries, including medical and electronics.

5.2.6 Customization and Rapid Prototyping:

Additive manufacturing (3D printing) is increasingly utilized for mold prototyping, offering faster iterations and customization options. This capability accelerates the product development cycle and facilitates the testing of various design iterations.

5.2.7 Continuous Innovation:

The injection machine mold industry continues to evolve, driven by ongoing research, material advancements, and customer requirements. Mold designers and manufacturers must remain open to innovation and adopt new techniques to stay competitive in the market.

5.2.8 Low Production Cost

Scrap material is used for the production of plastic parts that reduces production cost.

In conclusion, the injection machine mold serves as the backbone of the injection molding process, impacting the quality, productivity, and versatility of plastic product manufacturing. By embracing technological advancements, sustainable practices, and innovative designs, the injection machine mold industry will continue to shape the future of manufacturing, contributing to the development of efficient, high-quality, and environmentally conscious plastic products.

5.3 Safety Precautions

Personal Protective Equipment (PPE): Wear appropriate PPE, such as safety glasses or goggles, face shields, gloves, and protective clothing, to protect against potential hazards like molten plastic, flying debris, or chemical exposure.

Machine Safety: Familiarize yourself with the specific safety features and procedures of the injection machine. Ensure that emergency stop buttons, safety interlocks, and guards are in place and functioning correctly. Follow lockout/tagout procedures when performing maintenance or servicing the machine. Training and Knowledge: Receive proper training on the operation, maintenance, and safety procedures of injection machine molds. Understand the potential risks and hazards associated with the process, as well as the safe handling of materials, tools, and equipment.

Material Handling: Handle plastic resins and additives safely. Follow proper storage and handling guidelines to prevent exposure to hazardous chemicals or substances. Be aware of the potential for fires or explosions in case of mishandling or overheating of materials.

Mold Inspection: Regularly inspect the injection machine mold for any signs of damage, wear, or defects. Replace or repair any damaged parts promptly to avoid potential hazards during operation. Tool and Equipment Safety: Use tools and equipment properly and safely. Ensure that they are in good working condition and appropriate for the task at hand. Avoid using damaged or malfunctioning tools and equipment.

Heating and Cooling Systems: Take precautions when dealing with the heating and cooling systems of the injection machine mold. Follow the manufacturer's guidelines for proper temperature control, and be cautious of hot surfaces or fluids to avoid burns or scalds.

Housekeeping: Maintain a clean and organized work area to minimize tripping hazards and prevent clutter around the injection machine. Clean up any spills or debris promptly to maintain a safe working environment.

Ventilation: Ensure adequate ventilation in the work area to prevent the accumulation of fumes or gases that may be released during the injection molding process.

Emergency Preparedness: Know the location and proper use of emergency equipment, such as fire extinguishers and first aid kits. Be familiar with emergency procedures and evacuation routes.

SUSTAINABLE DEVELOPMENT GOALS SDGs

Our project falls in some of the following SDGs.

SDG 8: Decent Work and Economic Growth

SDG 9: Industry, Innovation & Infrastructure | Access to Insurance Initiative.

List of References

1. Hamad K, K. M. Practical Applications of Taguchi Method for Optimization of Processing Parameters for Plastic Injection Molding: A Retrospective Review., & 98:2801–12. (n.d.).

2. Ng Chin Fei,1Nik Mizamzul Mehat,2and Shahrul Kamaruddin.C. Gornik, D. r. (n.d.).E. Friederich, M.
3. f. (n.d.).M.S. Huang, C. p.-t.-p. (n.d.).Practical Applications of Taguchi Method for Optimization of Processing Parameterfor Plastic Injection Moulding.

4. M.A. Elbestawi, L. Chen, C.E. Becze, T.I. El-Wardany, Highspeed milling of dies and molds in their
5. hardened state, Annals of the CIRP 46 (1) (1997) 57–62.

6. P. Fallbohmer, C.A. Rodriguez, T. Ozel, T. Altan, High-speed machining of cast iron and alloy steels for die and mold manufacturing, J. Mater. Process. Technol. 98 (2000) 104–115. C.E. Becze, P. Clayton, L. Chen, T.I. El-Wardany, M.A. Elbestawi, High-speed five-axis milling of hardened tool steel, Int. J. Machine Tools Manuf. 40 (2000) 869–885. 113 (2001) 360–367.]

7. P. Be´zier, Courbes et surfaces, in: Mathe´matiques et CAO, vol. 4, Herme`s, Paris, 1984. J. Seon Hwang, T. Chien Chang, Three-axis machining of compound surfaces using flat and filleted end mills, Computer-Aided Design 30 (8) (1998) 641–647. L. Chih-China.

8. Tang, S.H., Tan, Y.J., Sapuan, S.M., Sulaiman, S., Ismail, N. and Samin, R., 2007, "The use of Taguchi method in the design of plastic injection mould for reducing warpage". Journal of Material Processing Technology, 182, pp.418-426

9. Deng YM, Britton GA, Lam YC, Ma YS (2002) Feature-based CAD/CAE integration model for
10. injection-moulded product design. Int J Prod Res 15:3737–375.

11. Deng YM, Britton GA, Lam YC, Ma YS (2001) A feature-based CAD-CAE integration model for injection molded product design. Proceedings of the 16th International Conference on Production Research, 2001, Prague, 234–241 7. Deng YM, Britton GA, Lam YC, M

12. Tucker, Charles L. HI, Fundamentals of Computer Modeling for Polymer Processing, Hanser, Munich, 1989. Boothroyd, Geoffrey with Peter Dewhurst and Winston Knight, Product Design for Manufacture and Assembly, M. Dekker, New York. 1994.

13. Q. Wang, M. Zhen, Z. Wu, Y. Cai, T. Kuo-Ming, and L. Jun-Kai, "Effect of process parameters on cavity pressure in injection molding," AIP Conference Proceedings, vol.

1820, no. 1, Article ID 050005, 2017. H. Hassan, "An experimental work on the effect of injection molding parameters on the cavity pressure and product weight," The International Journal of Advanced Manufacturing Technology, vol. 67, no. 1– 4, pp. 675–686, 2013.

14. Ch Hopmann, A. Reßmann, Self-optimizing in Injection Molding and the Problem at Compensating Viscosity Fluctuations, SPE ANTEC 2014, Las Vegas NV, US.

15. P. Kennedy, Flow Analysis of Injection Moulding, Hanser Gardner Verlag, Munich, 1995.

16. Hamad, K.; Kaseem, M.; Deri, F. Recycling of waste from polymer materials: An overview of the recent
17. works. Polym. Degrad.

18. Fu, G.U.; Hall, P. Performance Evaluation for Composites Based on Recycled Polypropylene Using PCA & CA. In Proceedings of the 6th International Conference Design and Manufacture for Sustainable Development, Hangzhou, China, 15–17 April 2013.

19. Latiff, A.A. Mechanical and Tribological Properties of Recycled Carbon Fiber Reinforced Polypropylene Composites. Master's Thesis, Universiti Teknikal Malaysia, Melaka, Malaysia, 2017.

20. Yu, M.; Wang, J.; Tian, P.; Sun, L.; Sun, K.; Ge, Z.; Huang, R. Evaluation of the Durability of Ligninreinforced Composites Based on Wheat Straw/Recycled Polypropylene Blends. BioResources 2019,14, 5683–5697.

21. Barghikar, H.; Mosaddegh, P.; Masoumi, M.; Ranjbar, M. The effect of packing phase and mold temperature on the directional warpage of spherical lenses using the injection molding process. SN Appl. Sci. 2019,1, 598.Macedo, C.; Freitas, C.; Brito, M.A.; Santos, G.; Faria, L.; Laranjeira, J.; Simoes, R. Influence of dynamic temperature control on the injection molding process of plastic components. Procedia Manuf. 2019,38, 1338–1346.

22. Chen, J.C.; Guo, G.; Wang, W.-N. Artificial neural network-based online defect detection system with in-mold temperature and pressure sensors for high precision injection molding. Int. J. Adv. Manuf. Technol. 2020,110, 2023–2033.

23. An effect of mold surface temperature on final product properties in the injection molding of high-
24. density materials. Polym. Bull. 2021,78, 2627–2644.

25. Rusdi, M.S.; Abdullah, M.Z.; Mahmud, A.S.; Khor, C.Y.; Aziz, A.; Ariff, Z.M.; Abdullah, M.K. Numerical Investigation on the Effect of Pressure and Temperature on the Melt Filling During Injection Molding Process. Arab. J. Sci. Eng.2016,41, 1907–1919.

26. Nam, J.S.; Baek, D.S.; Jo, H.H.; Song, J.Y.; Ha, T.H.; Lee, S.W. Lens injection moulding condition diagnosis and form error analysis using cavity pressure signals based on response surface methodology. Proc. Inst. Mech. Eng. Part B J. Eng. Manuf.2016,230

27. Hopmann, C.; Abel, D.; Heinisch, J.; Stemmler, S. Self-optimizing injection molding based on iterative learning cavity pressure control. Prod. Eng. 2017,11, 97–106.

28. Stemmler, S.; Vukovic, M.; Ay, M.; Heinisch, J.; Lockner, Y.; Abel, D.; Hopmann, C. Quality Control in Injection Molding based on Norm-optimal Iterative Learning Cavity Pressure Control. IFACPapersOnLine 2020,53, 10380–10387.

29. Huang, M.-S.; Ke, K.-C.; Liu, C.-Y. Cavity pressure-based holding pressure adjustment for enhancing
30. the consistency of injection molding quality. J. Appl. Polym. Sci. 2021,138, 50357.

31. Huang, M.-S.; Liu, C.-Y.; Ke, K.-C. Calibration of cavity pressure simulation using autoencoder and
32. multilayer perceptron neural networks. Polym. Eng. Sci. 2021,61, 2511–2521.

33. Chen, J.-Y.; Hung, P.-H.; Huang, M.-S. Determination of process parameters based on cavity pressure characteristics to enhance quality uniformity in injection molding. Int. J. Heat Mass Transf. 2021,180, 121788.

34. Poszwa, P.; Brz˛ek, P.; Gontarev, I. Influence of Processing Parameters on Clamping Force during Injection Molding Process; Springer: Cham, Switzerland, 2019.

35. Chen, J.-Y.; Zhuang, J.-X.; Huang, M.-S.2012

36. Monitoring, prediction and control of injection molding quality based on tie-bar elongation.J. Manuf. Process. 2019,46, 159–169.

37. Zhang, J.; Zhao, P.; Zhao, Y.; Huang, J.; Xia, N.; Fu, J. On-line measurement of cavity pressure during
38. injection molding viaultrasonic investigation of tie bar. Sens. Actuators A Phys. 2019,285, 118–126.

39. Zhao, P.; Wang, S.; Ying, J.; Fu, J. Non-destructive measurement of cavity pressure during injection molding process based onultrasonic technology and Gaussian process. Polym. Test. 2013,32, 1436– 1444.

40. Hassan, H. An experimental work on the effect of injection molding parameters on the cavity pressure
41. and product weight. Int. J.Adv. Manuf. Technol. 2013,67, 675–686.

42. Liang, X.H.; Liu, Z.L.; Pan, J.; Zuo, M.J. Spur Gear Tooth Pitting Propagation Assessment Using Modelbased Analysis. Chin. J.Mech. Eng. 2017,30, 1369–1382. ang SH, Tan YJ, Sapuan SM, Sulaiman S, Ismail N, Samin R, 2007. The use of Taguchi method in the design of plastic injection mould for reducing war page. J Mater Proc Technol, 182:418 26.

43. Fu MW,Fuh JYH, Nee AYC,1999. Undercut feature recognition in an injection mould design system. Computer Aided Design,;31:777 90.

44. Zhai M, Lam YC, Au CK, Liu DS,2005. Automated selection of gate location for plastic injection moulding processing. Polym PlastTechnol Eng ;44:229 42.