ABSTRACT

This has been an opportunity to study, design and manufacture of an Insert Moulding Tool of Base Cover Component. The material of Base Cover being ABS is used for supporting electronic equipment, for issuing tickets to passengers at KSRTC buses. Tool design protocol is on conventional lines, based on tested and tried norms developed through expertise and extensive experience over a number of years and reported in literature. Design of Core and Cavity and other parts to build the Mould Tool for normal working, Creation of detailed manufacturing drawing of all the parts of the tool, 3D Virtual Models is created for effective communication using the software Pro-E Wildfire2. The Core & Cavity were extracted using Pro-E Wildfire2. The 2D Drawings of the designed tool have been drawn using Auto CAD® 2002. Cost estimation procedure is carried out to predict the cost of a tool to be manufactured before it is actually manufactured. Fabrication is carried out starting from the design of electrodes, process planning and assembly procedures. The report is made complete by including design of electrodes, process planning, and assembly procedure and Component tryouts. Added to this the Component is checked for dimensional accuracy and a note on cost estimation of the tool is included.

Keywords: Intelligent design, cavity layout design, injection mould design, case-based reasoning, design support system.

1. Introduction

This project will use a plastic part whose model has been prepared in CAD software as a case study. It is anticipated that these concurrent tools will guide part designers and mold makers to decide the appropriate factors of design. There are various processes used to fabricate plastic products, ranging from household convenience packages to electronic devices and many others including the strongest products in the world, used in space vehicles, building structures etc. Proper process selection depends upon the nature and requirements of the plastic, the properties desired in the final product, the cost of the process, its speeds, and product volume. Some materials can be used with many kinds of processes; others require a specific or specialized machine. Numerous fabrication process variables play an important role and can influence product aesthetics, performances and cost. Worldwide plastic consumption is at least 125,000 million pounds (by weight). About 36% is processed by extruders, 32% percent is processed by injection moulding, 10% by blow moulding, 6% by calendars, 5% in coating, 3% in compression moulding, 2% in powder form, and 6% using other processes these percentages do not correlate with the number of machines used, for example there are three times more injection machines than extruders.

2. Literature Review

C. T. Wong, S. Sulaiman, N. Ismail and A.M.S. Hamouda: This paper presents the design of plastic injection mold for producing a plastic product. Before proceeding to injection machine and mould design, this part was analyzed and simulated by using mold flow. The analysis and simulation can define the most suitable injection location, material temperature and pressure for injection. The predicted weld lines and air trap were also found and analyzed.

R. Dubay, B. Pramujati and J. Hernandez: A new approach for controlling part cooling in plastic injection molding is developed using a Plastic Injection controller and coolant flow rate as the manipulated variable. The method uses an average part surface temperature within the mold as the set point parameter. A mechatronic system was developed for providing variable coolant flow rates.
E. O'Neill, C. Wilson and D. Brown: This paper seeks to outline the benefits of the solid model within the injection molding industry in greater detail. Solid modeling used as a fundamental tool for communicating geometric information. Solid modeling has become a core communication of concurrent engineering. The major benefit of CAE tools is reduced ‘time to market’. The reduced design time is an outcome which is a more important benefit of CAE is higher product quality.

3. Problem Statement

The dissertation work titled Design and Manufacture of a Single Cavity Mould for the component “Base Cover” using concurrent engineering design. However an attempt is made to compare concurrent design to traditional design for this product. The Base Cover Component is used for supporting electronic equipment, for issuing tickets to passengers at KSRTC busses. This component serves as a holding base cover for the entire equipment and it is produced from ABS material. The Component contains features like bosses, undercuts, holes etc., with close tolerances. After studying the geometry of the Component certain guidelines are made to go with the task.

3.1 Objectives

The main objective is to develop an improved understanding of the interrelation between component geometry, tool design, manufacturing process of the component, materials of the component, quality and productivity demand with constraints in cost of tooling, materials employed and fabrication.

Developing a design protocol for the tool to manufacture the component “Base Cover”, with the following design objectives

1. Design for component performance
2. Design for manufacturability i.e., to reduce manufacturing lead time
3. Design to minimize cost i.e., Design the tool as simple as possible.

3.2 Scope of the Work

The problem identified was approached scientifically by the following means:

- Analysis of the Component geometry which enables the choice of Parting surface, feeding (injection location), Ejection technique.
- Study of Physical, Mechanical & Processing properties of the moulding material primarily to enable the design of this tool.
- Design of the Injection mould tool.
- 3D Modeling of the tool designed using pro-E wildfire2 software for effective communication all the way along the design of process space to fabrication.
- Study of Physical, Mechanical and processing properties of the tooling materials and its influence with the moulding material.
- Creating detailed drawing of all the parts of the tool and its assembly drawing for manufacturing of the tool-using Auto CAD 2002.
- Process planning, Electrode planning
- Cost estimation is carried out to predict the cost of a tool to be manufactured before it is actually built.
- Follow up of tool manufacture and tool assembly. Which involves assisting the toolmaker in manufacturing and assembling the tool and to provide necessary data or clarifications from the design side? This helped in studying the actual procedure of a tool manufacturing and assembly
- Final tryout of the tool and correction of moulding defects.

3.3 Manufacturing Process for Base Cover Injection Moulding

Injection moulding is one of the most versatile processing methods adopted for manufacturing small clips to large crates. An injection moulding process generally is used for processing thermoplastics. In this process certain quantity of plastic material in granular form is heated & softened in one part of the machine & forced under pressure into the closed mould. The hot plastic will be cooled inside the mould until it solidified & retains the shape imparted to it. The injection moulding process is greatly preferred by designers because the manufacture of parts of complex shape in three dimensions can be more accurately controlled and predicted with injection moulding than with other processes.

The injection mould is an assembly of parts containing within it an impression into which plastic material is injected and cooled. It is the impression, which gives the moulding its form. The impression may therefore, be the part of the mould, which imparts shape to the moulding.
3.4 Feed System
It is necessary to provide a flow-way in the injection mould to connect the nozzle (of the injection machine) to the impression. This flow-way is termed the “feed system”. Normally the feed system comprises of a sprue, runner & gate.

3.4.1 Runner Size
When deciding the size of the runner, the following factors are to be considered
- The wall section & volume of the moulding.
- The distance of the impression from the main runner or sprue = flow length
- Runner cooling considerations.
- The plastic material to be used.
- Cycle time
The following empirical formula used to determine the size of the runner

\[
d_r = \frac{\sqrt{W \times L_r}}{3.7}
\]  

Where, \(d_r\) = diameter of the runner (mm)
\(W\) = weight of the component with losses (gms)
\(L_r\) = length of the runner (mm)

The half round diameter of the runner

\[
D = \sqrt{2 \times d_r}
\]

Where, \(D\) = Half round runner diameter (mm)
\(d_r\) = diameter of the runner (mm)

3.4.2 General Points Considered In Cooling Circuit Design
1. The cooling medium must not be allowed too close to the impression (closer than 16mm), since this results in cold spots and may lead to moulding faults such as weld lines. Care, too, must be taken to prevent the cooling holes from being so close to the mould surface as there is danger that the cavity pressure of the plastic material could cause localized collapse of the tool.
2. The amount of cooling provided must be sufficient to maintain the temperature of the tool when the latter is running at the maximum production.
3.4.3 Mould Construction

![Assembled view of the Tool](Source: Developed for the research)

The injection mould is an assembly of parts containing within it an *impression* into which plastic material is injected and cooled. It is the *impression*, which gives the moulding its form. The impression may therefore, be the part of the mould, which imparts shape to the moulding. It can be seen Fig 2.41 that the various mould parts fall naturally into two sections or halves. Hence, that half attached to the stationary platen of the machine is termed as fixed half and the other half of the mould attached to the moving platen of the machine as the mould half.

4. Methodology

1. Component study and 3D-solid modeling of the Component
2. Design of the Tool with proper selection of tooling materials
3. 3D solid modeling of the Tool
4. Fabrication of the Tool
5. Analysis of the Tool
6. Tool assembly and tryouts
7. Quality inspection of the manufactured Component.

4.1 Component study and 3D-solid modeling of the Component

![Base Cover Component](Source: Developed for the research)

In this section, the basic function of the component, important features, geometrical data of the part that is to be produced are discussed.
4.2 Selection of Tool Materials

<table>
<thead>
<tr>
<th>Sl. No</th>
<th>TOOL ELEMENTS</th>
<th>TYPE OF MATERIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Top plate, Bottom plate, Cavity &amp; Core housing, Core back plate, Ejector plate, Ejector back plate, Spacer block, Locating ring</td>
<td>C45</td>
</tr>
<tr>
<td>2</td>
<td>Cavity insert, Core insert, Splits, Form pins</td>
<td>T35Cr5MoV1</td>
</tr>
<tr>
<td>3</td>
<td>Sprue bush, Sprue puller, Wedge block, Wear plate, Guide ways, Stop pin, Finger cam pin, Return pin, Ejector pin</td>
<td>T110W2Cr1</td>
</tr>
<tr>
<td>4</td>
<td>Guide pillar, Guide bush, Centering bush, Ejector Guide pillar, Ejector Guide bush</td>
<td>17Mn1Cr95</td>
</tr>
</tbody>
</table>

5.3D MODELING OF TOOL

![Fig A & B: Fixed Half & Moving Half of the Tool](Source: Developed for the research)

5.1 Fixed Half & Moving Half Of The Tool In Opened Condition & Closed Condition.

![Fig A & B: Fixed Half & Moving Half Of The Tool In Opened Condition & closed condition.](Source: Developed for the research)

6. Tool Manufacturing, Assembly & Tryout.

In the process of manufacturing, the raw material is converted into the finished tool. Once the tool is designed, the next step is to manufacture the tool. Injection mould tool could be manufactured in many ways. The first & the basic process that is considered for manufacturing the tool are by conventional process. In some cases, it is not possible to make the tool by conventional methods because of the complex profiles that are involved in the mould.

During tool manufacturing the following procedure is followed

- Process planning
- Electrode planning
- Heat treatment for specific parts
- Manufacturing.
Fig A & B respectively shows the Sparking (EDM) of Cavity Insert & Core Insert used in this Project work.

[Source: Developed for the research]

6.1 Assembled View Of Tool.

Fig. Assembled View Of Tool [Source: Developed for the research]

7. Conclusion

The dissertation work carried out on Base Cover Component indeed gave me a wonderful experience in aspects of design and fabrication of an insert Mould. A successful execution of any activity requires a good co-ordination from many departments. The design activity starts with an abstract idea of the tool, which then translates in to full - fledged production drawing. It is equally important that the tool in manufactured considering all the intricacies of the process. The whole exercise is to finally get defect free components conforming specification with built in tool quality in the most economical way. The Sink mark found in the rib portion from trial component was avoided by increasing the Holding pressure & Holding time of the machine.

The project undertaken has been successfully accomplished by carrying out a design exercise, choosing a steel alloy for the mould with good balance of strength & manufacturability. Engineering the gating to promote soundness. Satisfaction technical, economic and delivery schedule requirements. The design of the Injection mould has been approved. The tool is manufactured in - house. The trial, proving of the tool & inspection of the components has been done in – house.

8. Scope for Future Work

- The feasibility study to replace conventional feeding with the hot runner or hot tip systems, which can greatly reduce the scrap percentage (100% with employing hot runner and 30-50% by using a hot tip), and effect of these on the quality of the moulded component can be studied.
- Use of Non – ferrous metallic material such as Beryllium – Copper alloy for mold making should be studied (due to high thermal conductivity, cycle time can be further optimized)
- Fatigue analysis can be carried out using FEA software to determine the life of the mold. The results can be used for, further optimization of the tool design.
- As for as the design of injection moulds is concerned lot of research work has to be carried out. Scientific design techniques are to be incorporated by eliminating trial and error basis.
References


[5]. https://www.plasticsportal.net/wa/plasticsEU/portal/show/content/injection_troubleshooter/


[7]. Menges/Mohren “How to make Injection Moulds”

[8]. Injection molding processing and selection of injection moulding machine guide. PDF from www.honeywellplastics.com/

[9]. Technical papers by Prabodh C.Bolur “Technological solutions for total quality in injection moulding of plastics”, “Understanding selection of injection moulding machine”.


AUTHORS

Mr. Naveen.R.Karki has completed B.E (IP) from Bapuji Institute of Engineering & Technology and pursing M.Tech. in Production Management from BVB College of Engineering & Technology. He has ten years of industry experience.

Mr. Chandrashekar.M has completed B.E.(Mech) from National Institute of Engineering and M.Tech. in Tool Design from Government Tool Room and Training Center. He has ten years of industry experience.

Dr. Prasanna Ravari has completed B.E.(Mech) from S.D.M.C.E.T, M.Tech. in Production Management from BVB College of Engineering & Technology and PhD (ME) V.T.U. Belgaum. He has sixteen years of teaching experience.