Design Development and Analysis of Injection Mould Tool for Divider Housing Median Marker Thermoplastic Component

C.P. Natesha, K. Ramesh Babu and K.S. Dinakar

Abstract--- This paper deals with the Design Development and Mould flow Analysis of injection mould tool for Divider Housing Median Marker Thermoplastic Component. Polycarbonate Thermoplastic material is used for the production of this component. PC is often viewed as the quintessential engineering thermoplastic, due to its combination of toughness, high strength, high heat deflection temperatures, and transparency. The tool has to be designed to produce a good quality component considering the ease of manufacturability, assembly and Positive ejection of the component within the minimum possible time and cost. The tool is of four cavity mould. The component has got flat opening, which requires two side cores. This asks for a finger cam in the tool construction. Analysis for filling, flow and cooling is carried out using Mold flow software, Mold flow is used for the best gate location. Any product to be manufactured invariably requires machines and tool. Tool design and development is a specialized and critical area. The tool design should match the machine specification and should be accurate and economical for successful life of a product.

Keywords--- Tool Design, Injection Mold, Side Core, Finger Cam, Mold Flow

I. INTRODUCTION

ONE of the most common methods of converting plastics from the raw material form to an article of use is the process of injection moulding. This process is most typically used for thermoplastic materials which may be successively melted, reshaped and cooled. Injection moulded components are a feature of almost every functional manufactured article in the modern world, from automotive products through to food packaging. This versatile process allows us to produce high quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology. Optimal setting up of injection molding process variables plays a very important role in controlling the quality of the injection molded products [3]. Mould design and fabrication is a costly and high technology process because it uses science-based computer-aided engineering (CAE) software to analyse and simulate the plastic parts, computer-aided design (CAD) software to design the complicated plastic product and computer-aided manufacturing (CAM) to do the programming fabrication to run the computer numeric control (CNC) milling [4].

1.1. Injection Moulding Cycle

The time from the moment the mould is closed for one injection, until it is closed for the following injection is termed as cycle time. Typical cycle time ranges from 20 sec to 60 sec which leads to a high production rate with good dimensional tolerance control.

The injection moulding process basically involves three phases

- Filling phase
- Pressure phase
II. COMPONENT [DIVIDER HOUSING MEDIAN MARKER]

2.1. Component Details
Component name: Divider housing median marker
Component material: Polycarbonate (Laxan)
Shrinkage: 0.6 %
Component weight: 106 grams (Single component)
Max. wall thickness of the component: 80 mm
Moulding type: 4 Cavity injection mould tool
Tonnage required: 250 tonnage capacity
Max. Temperature: 121°C
Specific gravity: 1.20 g/cm³
Projected area of component: 685.53 cm² (From CAD model).
Projected area of component including feed system: 689.80 cm² (From CAD model).

2.2. Selection of Injection Molding Machine
The minimum machine tonnage required is 207 tonnes.
It is suggested that the available machine is Toshiba 250 Machine.

2.3. Criticality in the Component
- Chances of cladding between the honeycomb structure side core and the honeycomb structure of the component.
- Ejector pin marks on the face of the component.
- Dimensions maintain on the ribs is important.
- Proper alignment of the housing.

III. DESIGN – CAD ASSEMBLY
The designing was carried out by using Top solid 2009 [Missler] software.

![Die Set Assembly](image)

Figure 1: Die Set Assembly

The various mould parts fall naturally into two sections or halves. Hence, that half attached to the stationary platen of the machine is termed the fixed half. The other half of the mould attached to the moving platen of the machine is known as moving half [1].
3.1. Side Cores

Side core is a local core which is normally mounted on right angles to the mould axis for forming a hole or recess in the side face of the moulding [1].

Advantages of side cores,

- The mould is make simple
- Reduce the moulding cost
- Ease of operation

3.2. Finger Cam Actuation

Figure 4: Side Cores

Figure 5: Finger Cam

Figure 6: Cross Section of Die Set
In this system, hardened circular steel pins, termed finger cams, are mounted at an angle in the fixed mould plate. The splits [side cores] mounted in guides on the moving mould plate, have corresponding angled circular holes to accommodate these finger cams [1].

<table>
<thead>
<tr>
<th>Mould Parts</th>
<th>Mould Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top plate</td>
<td>C45</td>
</tr>
<tr>
<td>Cavity plate</td>
<td>Orvar Supreme</td>
</tr>
<tr>
<td>Core plate</td>
<td>Orvar Supreme</td>
</tr>
<tr>
<td>Core back plate</td>
<td>C45</td>
</tr>
<tr>
<td>Spacers</td>
<td>C45</td>
</tr>
<tr>
<td>Ejector plate</td>
<td>C45</td>
</tr>
<tr>
<td>Ejector back plate</td>
<td>C45</td>
</tr>
<tr>
<td>Bottom plate</td>
<td>C45</td>
</tr>
</tbody>
</table>

IV. MOLD FLOW ANALYSIS

Mold flow Plastics Insight software represents the most comprehensive suite of definitive tools for simulating, analyzing, optimizing, and validating plastics part and mould designs. Mold flow is an outstanding predictive engineering tool. It’s a simulation tool which can identify potential problems or address uncertainty. The scientific approach used will save both time and money. To eliminate uncertainty and to insure the design can be molded successfully. Mold flow analysis is a “must have” for reducing costs and optimizing productivity.

Mold flow Plastics Insight (MPI) software which is an integrated suite of analysis tools that utilize CAD files and apply advanced Finite Element Analysis (FEA) techniques to quickly and easily enable a virtual “what if” design environment before initiating mould construction. MPI provides in-depth part/mould design and process parameter optimization.

Mold flow is a plastics flow simulation product, which allows mould designers to optimize single cavity, multi-cavity and family mould layouts, eliminate potential downstream problems and avoid costly rework and manufacturing delays. Most polymers exhibit two regimes of flow behavior, Newtonian and shear-thinning. Newtonian flow occurs at low shear rates, but with increasing shear, the viscosity tends to fall away in what is termed shear-thinning behavior. Viscosity also decreases with increasing temperature [2].

The advantages of using Mold flow for simulation:

- Design and analyze virtually all types of hot and cold sprue and runner systems.
- Obtain automatically sized runners to balance flow in multi-cavity and family moulds.
- Lower development and production costs.
- Obtain practical, results-specific advice on improving the part and mould design.
- Visualize the orientation of the plastic to aid in maximizing part strength, especially in the vicinity of weld lines.
- Communicate valuable information to plastics part designers, mould makers and manufacturing engineers.
- Identify the most suitable plastics material candidate.
- Identify and eliminate cosmetic issues such as sink marks, weld lines and air traps.
- Determine the best gate locations and the optimum number of gates.
- Predict estimated cycle time, clamp tonnage and shot volume.

4.1. Mold Flow Analysis

- Plastic Filling Analysis.
- Warp Analysis.
- Mould Cool Analysis.
- Gate Location Analysis.
4.2. Data Input
- Material = Poly carbonate [Laxan]
- Injection location = as per the best gate location analysis
- Mold temperature = 80°C
- Melt temperature = 280°C
- Type of analysis = Fill
- Type of mesh = Fusion
- Feed system = Runner, Gate, Sprue according to mold design
- Cooling system = According to mold design

4.3. Best Gate Location
The Gate Location result rates each place on the model for its suitability for an injection location. The most suitable areas, colored blue, are rated as best, and the least suitable areas of the model, colored red, are rated as worst.

4.4. Fill Time
This result shows the flow path of the plastic through the part by plotting contours which join regions filling at the same time. These contours are displayed in a range of colors from red, to indicate the first region to fill, through to blue to indicate the last region to fill. A short shot is a part of the model that did not fill, and will be displayed as translucent. By plotting these contours in time sequence, the impression is given of plastic actually flowing into the mould.
4.5. Temperature at Flow Front

The flow front temperature result uses a range of colors to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red). The colors represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.

4.6. Air Traps

The air traps result is generated at the end of a Mid-plane or Fusion flow analysis, and shows a red line wherever an air trap is likely to occur. This may be where the melt stops at a convergence of at least 2 flow fronts, or at the ends of flow paths.

Remedies: Balance flow paths, balance runners, changing the runner system can alter the filling pattern in such a way that the last-to-fill areas are located at the proper venting locations, proper venting.

4.7. Weld Lines

When a weld line forms, the thin frozen layers at the front of each flow path meet, melt, and then freeze again with the rest of the plastic. The orientation of the metal at the weld is therefore perpendicular to the flow path. This result indicates the presence and location of weld and meld lines in the filled part model. These are places where two flow fronts have converged. The presence of weld and meld lines may indicate a weakness or blemish. Increasing the injection pressure and adjust cooling water temperature in that zone is likely to avoid weld lines and give us good component. As this problem was occurred in the similar designs and those problems are overcame by slight adjustment in the process parameters. Single Gate Unidirectional Filling - Adjacent flow weld-line/meldline formed.

Multi Gate Bidirectional Filling - Cold weld-line formed at the joint [6].
V. CONCLUSION

Injection molding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost\(^5\). The presented work, deals with the design development and analysis of injection mould tool of a Divider housing. CAD/CAE technology facilitates the use of numerically controlled machining technology in manufacturing of mould. In turn, this reduces number and complexity of manual setup operations. The designing was carried out with Top solid, and analysis using Moldflow. Analysis programs can be of great assistance in enhancing quality of first time mould designs and also they provide a great means to define the moulding extremes of acceptable process conditions, before the mould is put to use on the production floor. Location of gates is quite difficult in family moulds, using Mold flow analysis best gate location is selected. Also Mould flow analysis is carried out to ascertain the possible variations in the plastic flow and other process variables. The results of the numerical simulations depend mostly on input data accuracy. Accuracy means here the knowledge of detailed description of the real materials with use of model as well as the material and process parameters\(^2\). The results were interpreted and necessary actions were taken to get a quality product.

REFERENCES