International Conference on Innovative Research in Engineering, Management and Sciences
International Journal of Scientific Research in Computer Science, Engineering and Information Technology

Mold Fill Analysis of Injection Molding Tool

Chetan Kumar D S [1], Shivaprakash S [2]
[1, 2] Senior Assistant Professor, Department of Mechanical Engineering, New Horizon College of Engineering Bengaluru, India

ABSTRACT

Injection mold Pro-E part advisor which is 3D solids- based plastics flow simulation tool used to predict and eliminate potential manufacturing problems during the design stage itself. Mold flow addresses the broadest range of manufacturing issues and design geometry types associated with injection molding process.
Keywords : Confidence of Fill, Pressure Drop, Fill Time, Fill Time Injection Pressure, Flow Front Temperature, Weld Line, Air Traps

I. INTRODUCTION

One of the greatest advantages of the plastics injection molding process is that parts of extremely complex geometry can be produced “net shaped” that is, once a part is molded, cooled and ejected from a mold, it is in the form required for the next step of manufacturing process. Net-shaped parts can be produced in high volumes, in a relatively short amount of time, and at a relatively low cost. However, the many potential problems associated with manufacturing such complex shapes can result in increased time to make and have a negative impact on profitability.

To avoid the high costs and time delays associated with problems discovered in the manufacturing environment, it is necessary to consider the combined effects of part geometry, material selection, mold design, and processing conditions on the manufacturability of a part.

This paper describes features of Injection mold Pro-E part advisor which is 3D solids-based plastics flow simulation tool used to predict and eliminate potential manufacturing problems during the design stage itself. Mold flow addresses the broadest range of manufacturing issues and design geometry types associated with injection molding process.

II. OBJECTIVES

- Analyze 3D CAD solid models directly, without having to manually create an analysis mesh.
- Evaluate the manufacturing feasibility of every part design by iteration.
- Identify the most suitable plastics material.
- Optimize the part wall thickness to achieve uniform filling, minimum cycle time and lowest part cost.
- Identify and eliminate defects such as sink marks, weld lines and air traps.
- Obtain practical, results-specific advice on improving the part design.
- Visualize the orientation of the plastic to aid in maximizing part strength, especially in the vicinity of weld lines
- Determine the best gate locations for a given part design
- Estimate the impact of changes to the part design or material selection on the overall part cost.

- Communicate key design analysis aspects to the mold designer through automated, Web-based, HTML project reports.

III. MOLDING PARAMETERS USED FOR ANALYSIS

The model was analyzed using theoretical parameters Molding temperature: 40°C
Melt temperature: 270°C
Maximum Injection Pressure: 100 MP
RESULTS
The results of various predictions of Mold flow Part Advisor are discussed below.

CONFIDENCE OF FILL
The confidence of fill result displays the probability of a region within the cavity filling with plastic. The confidence of fill result is calculated from the pressure drop and temperature results. Whenever we have a part which has yellow or red sections in the confidence of fill result, we should look carefully at the pressure and temperature results to determine where the problem is. The interaction between pressure and temperature is as shown in the plot below.

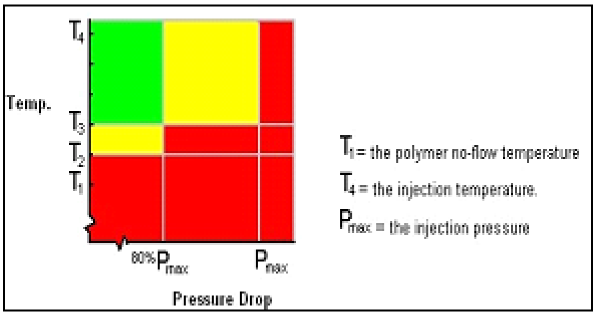

Fig 1. Temperature and Pressure plot

The temperatures on the plot are defined using the no-flow temperature for the polymer T1 and the temperature at the injection point T4 the other two temperatures are derived from the difference between T1 and T4. This difference is divided into five equal parts as shown on the plot. This means that T2 is 20% higher than T1 and T3 20% higher again, both relative to T4.

Therefore, high confidence of fill occurs when the pressure is less than 80% and the temperature is between T3 and the injection temperature T4.

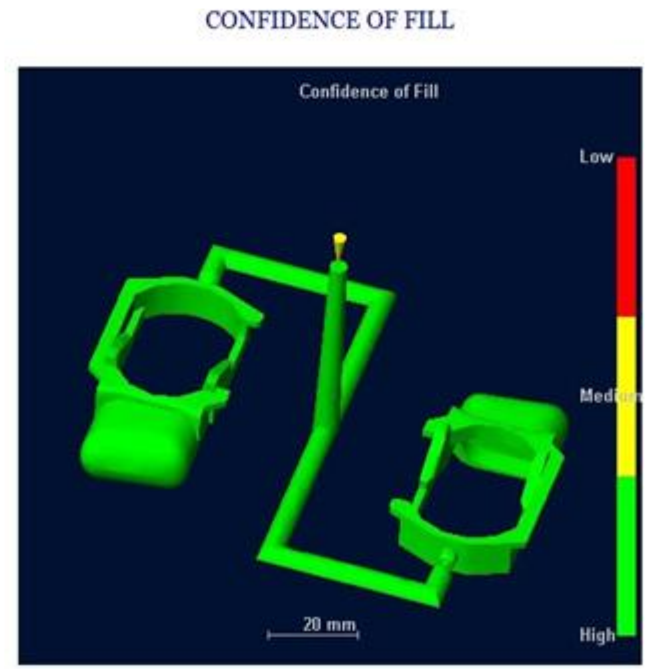

Fig 2. Results obtained for Confidence of fill analysis

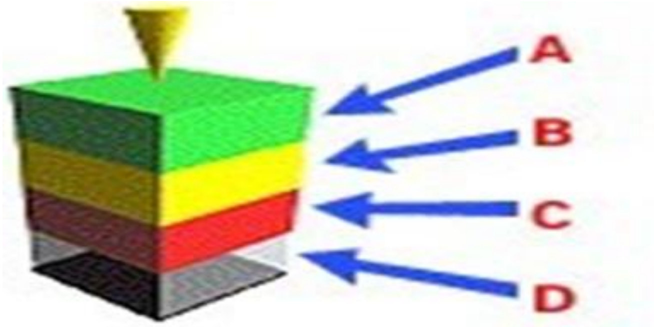

Fig 3.Colors displayed in the confidence of fill result are interpreted in the following manner:

A- Will definitely fill.
B- May be difficult to fill or may have quality problems. C- Will be difficult to fill or will have quality problems. D- Will not fill.

PRESSURE DROP
The pressure drop result uses a range of colors to indicate the region of highest pressure drop (colored red) through to the region of lowest pressure drop (colored blue). The color at each place on the model represents the drop in pressure from the injection location to that place on the model, at the moment

that place was filled. That is, the pressure required to force material to flow to that point.

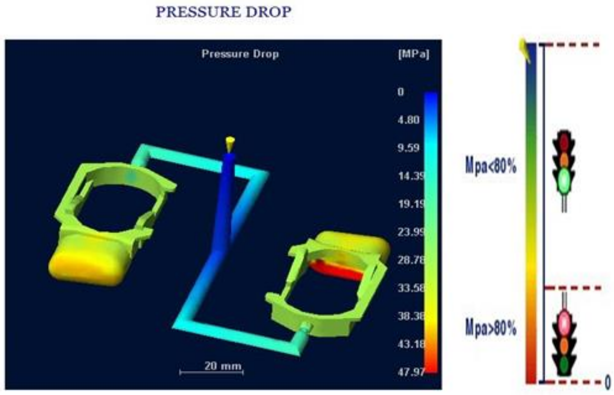

Fig 4. Results obtained for pressure analysis

The pressure drop is one factor used to determine the confidence of fill result. If the pressure drop is greater than 80% of the current value set for Maximum Injection Pressure Limit, in the Molding Parameters dialog, then this causes a yellow confidence of fill. When the pressure drop reaches 100% of the current Pressure Limit setting, the confidence of fill for this area is red. it is clear that the pressure drop is less than 80% so confidence of fill is 100%.

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 mold. For this particular component under consideration the fill time was found to be 2.99 sec.

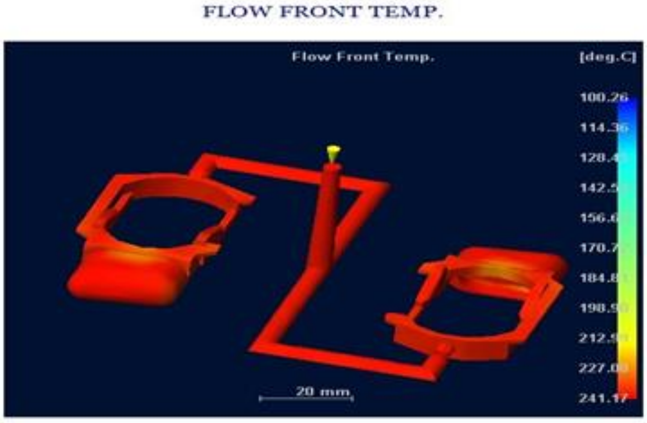

Fig 5. Time taken to fill the mold

IV. FILL TIME INJECTION PRESSURE

The injection pressure result uses a range of colors to indicate the region of lowest pressure (colored blue) through to the region of highest pressure (colored red).

The injection pressure can be used in conjunction with the pressure drop result. For example, even if a section of a part has an acceptable pressure drop, the actual injection pressure in the same area may be too high. High injection pressure can cause over packing.

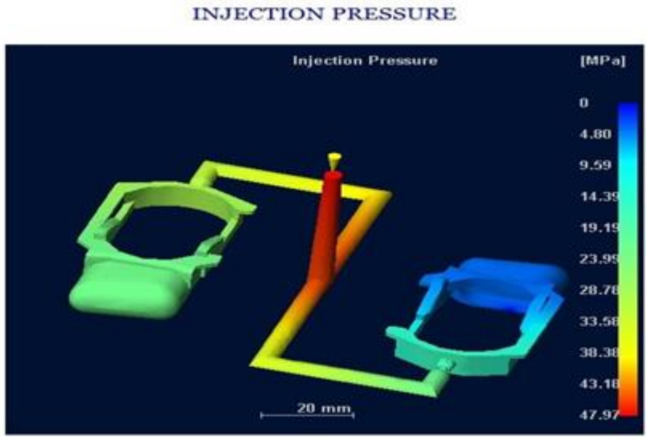

Fig 6. Plot Injection Pressure distribution with injection points

FLOW FRONT TEMPERATURE

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). If the flow front temperature is too low in a thin area of the part, hesitation or short shot may occur. If it is too low in an area where weld lines are present, the weld lines may appear worse.

In areas where the flow front temperature is too high, material degradation and surface defects may occur. We have to make sure that the flow front temperature is always within the recommended temperature range for the polymer we are using. The flow front temperature is one factor used to determine the confidence of fill result. Low melt temperatures will cause yellow or red confidence of fill results.

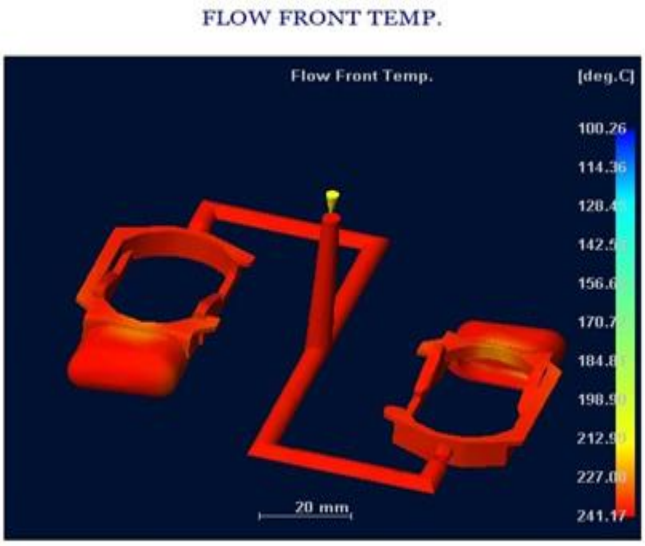

Fig 7. Plot of flow front temperature with double injection point

WELD LINE
Weld line forms, when 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 plastic at the weld is therefore perpendicular to the flow path. The weld line occurs where two flow fronts meet, and the polymer molecules are misaligned. It is the sharp difference in molecular orientation at the weld which causes the significant decrease in strength at this point.

Weld lines on a plastic part can cause structural problems and be visually unacceptable. Therefore weld lines should be avoided if possible. If it is not possible to remove a weld line, it should be positioned in the least sensitive area possible. This can be done by changing the polymer injection location or altering wall thicknesses to set up a different fill time. In a different fill time, flow fronts may meet at a different location and therefore the weld/meld line will move.

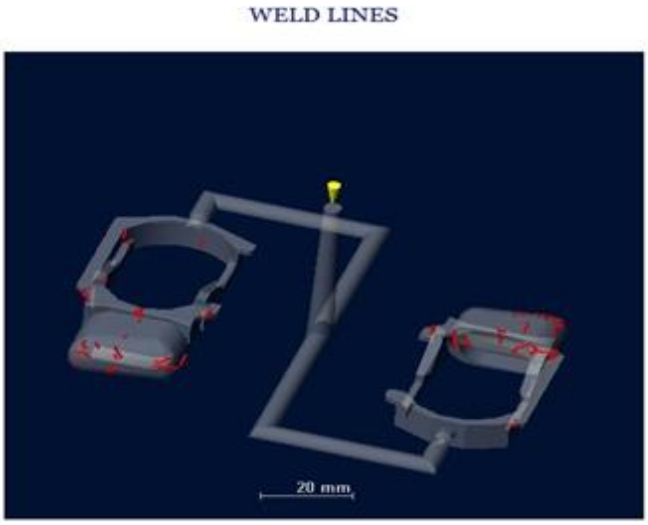

Fig 8. Weld location with two injection points

AIR TRAPS
The air trap result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped. The regions highlighted in the result are positions of possible air traps.

Air traps occur when converging flow fronts surround and trap a bubble of air. This normally happens where there is an unbalanced flow path. The racetrack effect can often give rise to flow fronts racing ahead along thick paths trapping a pocket of gas. In the following example the plastic races along the thick edge trapping a pocket of gas near the corner of the part. Another problem that can cause air traps is hesitation, where the plastic slows while traveling along a thin path. In the following example the top wall is thin, slowing the flow of plastic along this path. As a result, a pocket of air is trapped in the center of the front wall.

Flow paths do not need the racetrack effect or hesitation to have unbalanced flow. In a part with uniform thickness, the physical length of flow paths may vary, and again air traps may occur. Even in the case of a part with balanced flow paths, air traps can still occur by inadequate venting at the end of the flow paths.

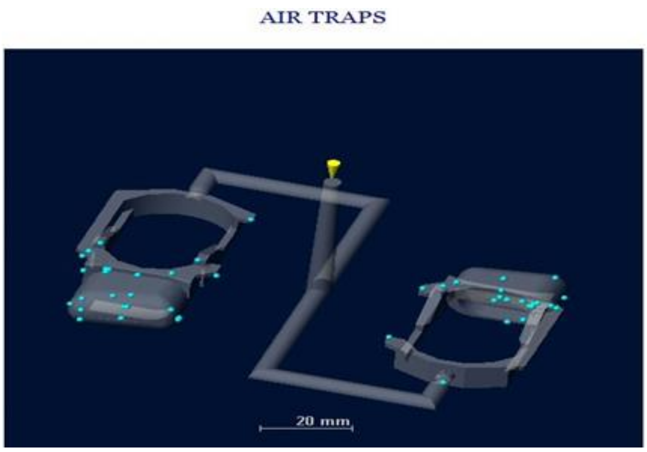

Fig11.Plot of Air entrapment

V. CONCLUSION

Above results of Part Advisor were very much useful to predict and eliminate potential molding problems before manufacturing of the tool. Also the results were helpful in better understanding of mold filling and flow. Some of the predictions and observations of mold fill analysis regarding air venting, weld lines, fill time, processing temperature and pressure, number of injection points and position, etc are driving parameters while designing the mold. For more detailed information about a model's molding behavior, Mold flow Plastic Insight analysis software can be used. Here we can address mold with gates, runner system, and cooling circuits to perform more in-depth analysis.

The overall quality of the molding is satisfactory at specified mold temperature and pressure (265 °C and 100 MPa) with single injection point.

VI. REFERENCES

[1] A. H. Dominighaus "Plastics for Engineers" Elsevier Applied Science Publishers,

[2] Domonic V Rosoto, Donald V Rosoto, Marlene G Rosoto, "Injection Molding Handbook", kluver academic publishers, 3rd Edition, 2000.

[3] Whelan, J. P. Goff, "Developments in Injection Molding", Elsevier Applied Science Publishers, Volume 1, 2, 3,

[4] A guide to injection moulding of plastic by Probodh C. Bolur

[5] Injection mould design fundamentals by a. b. glanvill and e.n.denton, Industrial Press Inc.,

[6] R.G.W. Pye, "Injection Mold Design", Longman Scientific and Technical Publication, 4th Edition, New Delhi,

[7] Gastrow Edited by K. Stoeckhert, "Injection Molds 108 Proven designs", Hanser publications,2 nd Edition