ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

Optimization of Toilet Brush Cup Mold Using Autodesk Mold Flow Advisor

[1]Joseph Irabodemeh Michael; [2] Kingsley Ikebudu. O.; [3]Arinzechukwu Hipolite Madukasi

[1]National Engineering Design Development Institute (NEDDI), Nnewi, Anambra State, Nigeria
[2,3]Mechanical Engineering Department, Chukwuemeka Odumegwu Ojukwu University Uli, Anambra State.

Publication Date: 2025/02/13

Abstract: The investigation of toilet brush cup mold was the main topic of this paper. The optimal gate location, fill time, time to attain the mold's ejection temperature, injection pressure, volumetric shrinkage at ejection, sink mark estimation, flow front temperature, and cooling time variance were all adjusted using Autodesk Mold Flow Advisor. This study examined the parameters established in a plastic injection mold along with the investigation of plastic flow behavior. The findings showed that minimal molding flaws might be produced in injection-molded components.

Keywords: *Optimization, Mold and Autodesk etc.*

How to Cite: Joseph Irabodemeh Michael; Kingsley Ikebudu.O.; Arinzechukwu Hipolite Madukasi (2025). Optimization of Toilet Brush Cup Mold Using Autodesk Mold Flow Advisor. *International Journal of Innovative Science and Research Technology*, 10(1), 2300-2306. https://doi.org/10.5281/zenodo.14862896

I. INTRODUCTION

For commercial application, plastics can be toughened and molded into a variety of shapes. Due to its strength, durability, ease of molding, and light weight, plastic finds utility in today's world. Amber, tortoise shells, and animal horns are examples of "natural polymers," despite the fact that plastics are considered a modern creation. These materials behaved a much as manufactured plastics do now, and they were frequently applied in ways that are comparable to those of produced plastics today. Injection molding, blow molding, compression molding, film insert molding, gas assist molding, rotational molding, structural foam molding, extrusion, and thermoforming are some of the procedures that can be used to create plastic products.

A plastic product is created through the technique of injection molding, which involves injecting plastic material into a mold. One manufacturing method for creating items out of thermoplastic is injection molding. After being heated and fed into an injection molding machine, the solid plastic material is forced into the mold. Plastic pellets or grains are fed into a heating chamber from a hopper during the injection molding process. The plastic is pushed into the heating chamber by a screw or plunger, softening it into a fluid state. The resin is pushed into a closed, cooled mold at the end of this chamber. The mold opens and the completed part is released when the plastic has solidified.

A. The Mold

The mold, also known as a tool, is constructed to precisely match the customer's specifications for the item or parts. Two mold halves usually make up the mold. Typically, the cavity and outside contour of the part are formed by one half of the mold. The cavity side is the name given to this portion of the mold. The inner shape of the part is formed by the projecting shape of the second mold half, which is known as the core. The shape of the object to be molded is defined by the hollow space created when the core is clamped against the cavity.

Typically, the cavity side of the mold is used to inject the plastic. To account for plastic shrinkage during cooling, the mold cavities are cut to larger proportions than the intended part dimensions. The part dimensions plus a shrink factor provided by the material manufacturer equals the cavity dimensions. Typically, two shrink factors are provided: one for dimensions perpendicular to the flow direction and one for dimensions in the flow direction. However, calculating shrinkage is not simple. In parts with complex geometries, it is frequently impossible to estimate the melt flow path, making it unclear the shrink factor to use. Process conditions also have an impact on part shrinkage. In order to achieve speedy manufacture of uniform products, heat must first be introduced into the material throughout the molding cycle and then removed as rapidly and consistently as feasible. Heat input is rather simple because the majority of contemporary injection molding equipment is screw machines. However, because plastic material has a low thermal conductivity and contains a lot of heat, it is challenging to remove heat from the material inside the mold.

B. Mold Cooling

Prior to ejection, cooling enables the plastic to solidify and achieve dimensional stability. A coolant that circulates via cored channels in the mold removes heat that the molten plastic has imparted to the mold. Heat removal efficiency is determined by coolant temperature and flow rate. Cooling the molded components evenly can be achieved by either employing the same flow rate of cooling medium throughout the mold but with varying cooling medium temperatures, or by cooling the mold with varying cooling medium flow rates in different places. In order to prevent flaws like uneven surface finish and changes in physical characteristics, the goal is to cool the components as rapidly and evenly as feasible.

The capacity to extract heat from the mold is also influenced by the layout of the cooling tunnels. Cooling will begin with the mold surfaces nearest the cored passageways. Variations in the distribution of mold temperatures will impact the reproducibility of molded components.

II. REVIEW OF PREVIOUS STUDIES

[1] investigated how asymmetric cooling affected injection-molded product warpage, particularly how packing pressure and mold temperature differential affected warpage for unfilled amorphous materials. More intriguingly, they demonstrated that when a strong holding pressure was applied, the plate geometry bent towards the cold side of the mold, causing the warpage to grow linearly with the applied temperature difference between the mold halves. It is well known and acknowledged that the plate bent toward the heated side at lower pressures. In a similar vein, the investigation of corner geometry with varying radii revealed that the angle deformation depended on both the packing pressure and the temperature differential, with an increased radius also suggesting heightened temperature sensitivity. Additionally, they compared their proposed model and the tests with the injection-molding program C-MOLD's warpage predictions. The model was successful in predicting the warpage's direction, but its absolute values were only roughly half of the measured values. The measurements revealed that this was only true for the lower packing pressures, contrary to C-MOLD's prediction that the warpage would always be oriented towards the hotter side of the mold. Numerous studies have examined the impact of temperature factors on warpage and shrinkage, and many of them have come to the conclusion that warpage forecasts are sensitive to even little changes in these parameters. [2] Found that altering the noflow temperature by 10° had a significant impact on the warpage prediction of a polystyren e disk.

Anisotropic and heterogeneous values utilized in the th reedimensional thermoelastic temperaturedependent model were less significant than this variation.

According to Jansen et al. (1999), changes in thermal c onductivity should have similar effects.

Another study by Jansen et al. (2002) examined how pr ocessing factors affected shrinkage for seven typical thermo plastic polymers.The impact of melt temperature was somew hat less significant than the holding pressure, which was the crucial factor.For each material in the investigation, the effe cts of mold temperature and injection speed on warpage wer e less pronounced and varied.The measured shrinkage for th e amorphous materials under all testing conditions was accur ately represented by a thermoelastic model that was created. Shrinkages were overestimated and the correlation was wea k for semicrystalline materials.

Another factor influencing the final part's size and shape is cavity deformation.[3] investigated the creation of a nonlinear mathematical model to analyze the injection molding machine's mold filling process. The Reynolds transport theorem, which describes the dynamics of polymer flow, is used to formulate the model. The temporary phenomenon of non-Newtonian fluids moving via a closed conduit can be used to simulate the mold filling process. When acrylonitrile-butadiene-styrene (ABS) is injected into a disk shaped mold, the nonlinear model is a reasonable approximation of the mold filling dynamics, according to a comparison between the experimental results and the theoretical simulation. Additionally, the dynamics of an injection molding machine's actuation system were examined. The findings show that the nonlinear model is also capable of accurately forecasting the actuation system's transient behavior.

4] examined how the cooling period and injection temp erature affected the ejection force of tiny PS and HDPE box es (122 x 50 x 27 mm).These findings display behavior that differs from previous research (104,107).The ejection force f irst falls at a specific injection temperature before rising as c ooling time increases.These findings implied the presence of an ideal cooling period during which the ejection force is re duced.The ideal cooling period often shifts toward a longer duration as the melt temperature rises.Wang et al. (2014) su ggested methods that link the ejection force to warpage, whi ch reduces with cooling time, and the temperature dependen cy of material stiffness, which increases with decreasing tem perature.In order to minimize waste and cycle time, the toile t brush cup mold was optimized in this study using Autodes k Mold Flow Advisor.

III. AIM AND OBJECTIVES OF STUDY

The aim of this work is to optimized toilet brush cup mold using Autodesk mold flow advisor

The objectives of this work are:

- To design injection mold using computer aided design software (solid works).
- To use Autodesk simulation mold flow advisor, to design the gating system, cooling system of the molding.
- To determine part moldability, detect and fix problematic zones.

ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

IV. RESULTS

Table 1: Input Details for Mold Flow Advisor

Item	Values
Injection pressure	29420Pa
Mould temperature	40ºC
Barrel temperature	250ºC
Injection time	7seconds

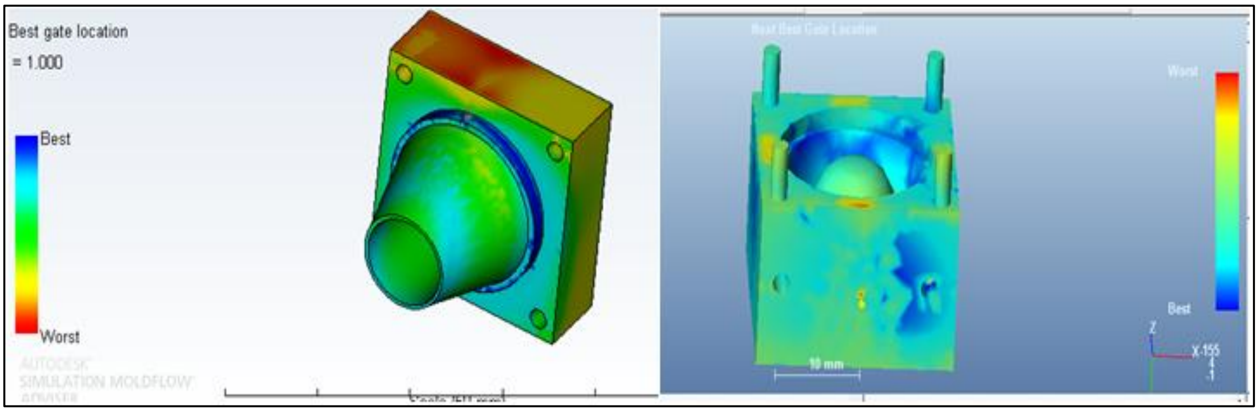

Fig 1: Best Gate Location

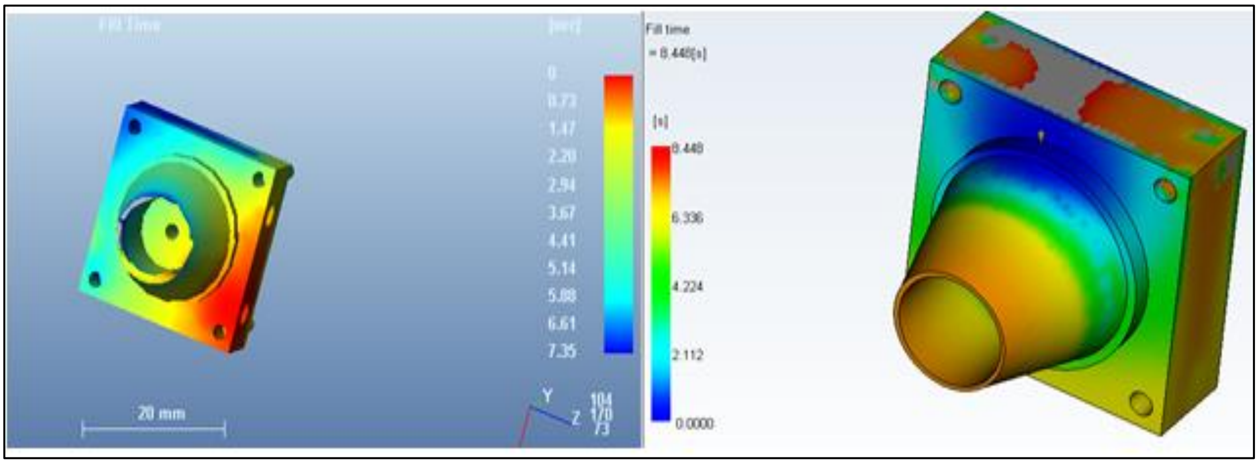

Fig 2: Fill Time Results of the Mold Flow Analysis

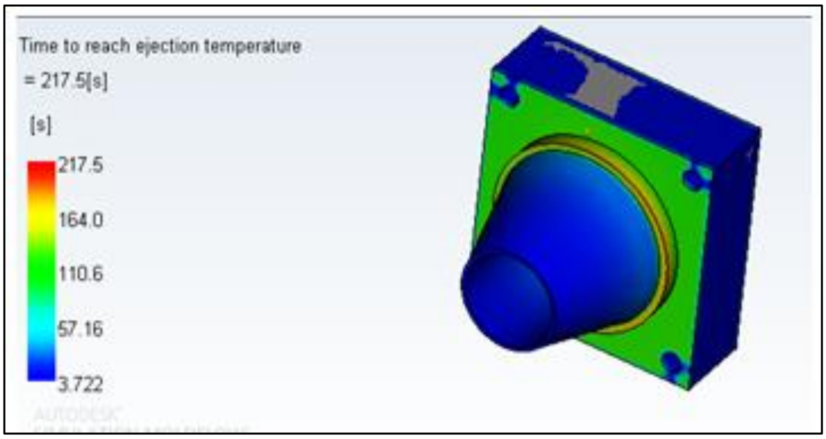

Fig 3: Time to Reach Ejection Temperature Results of the Mold Flow Analysis

ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

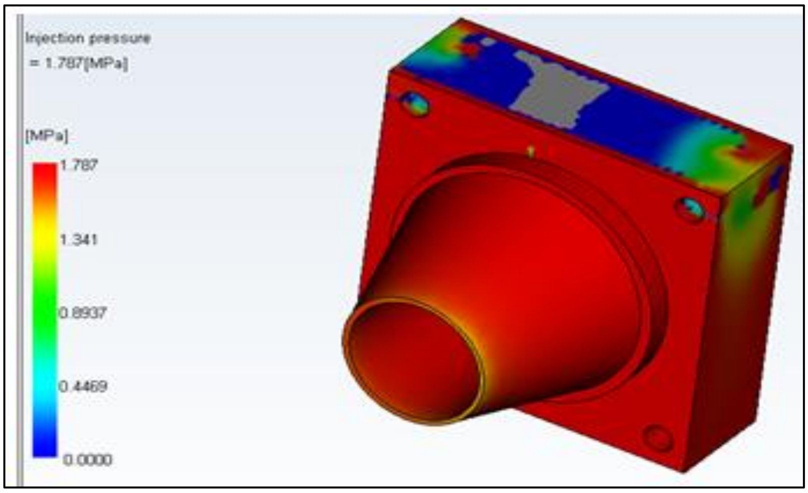

Fig 4: Injection Pressure Results of the Mold Flow Analysis

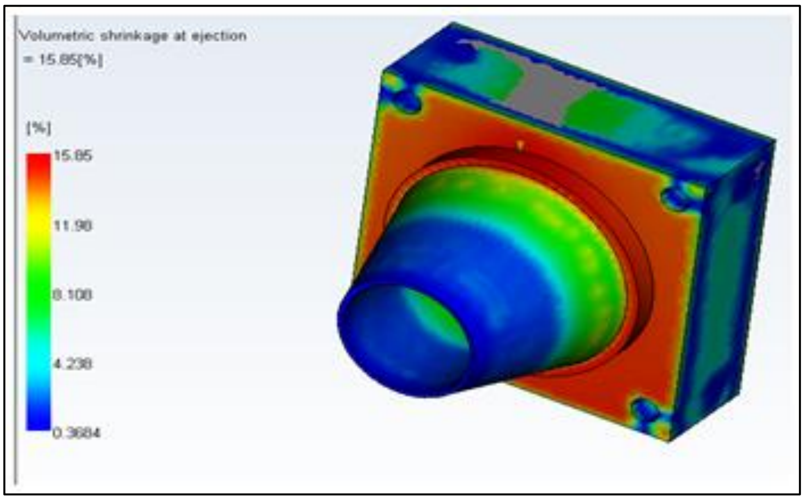

Fig 5: Volumetric Shrinkage at Ejection

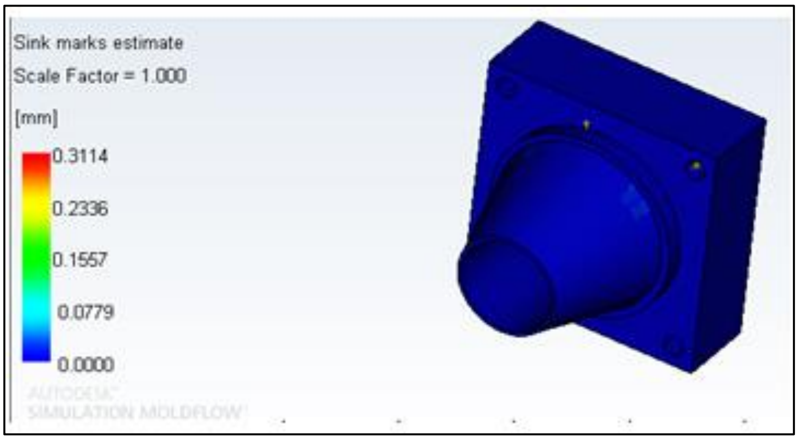

Fig 6: Sink Marks Estimate Results of the Mold Flow Analysis

ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

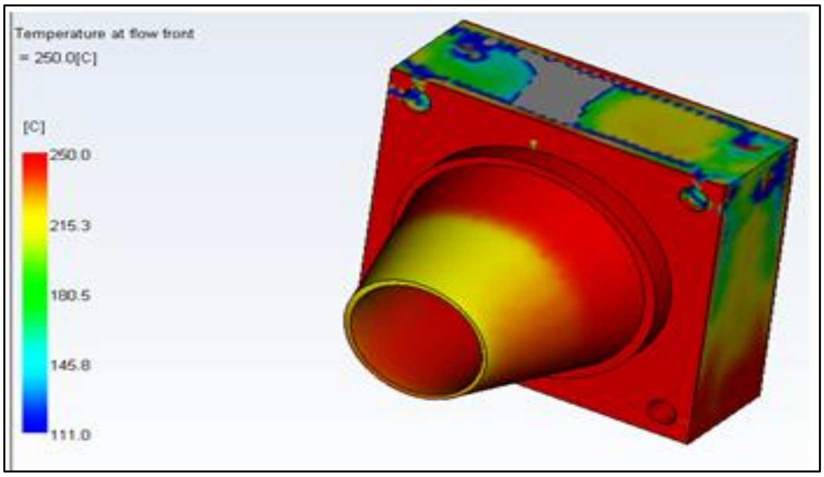

Fig 7: Temperature at Flow Front Results of the Mold Flow Analysis

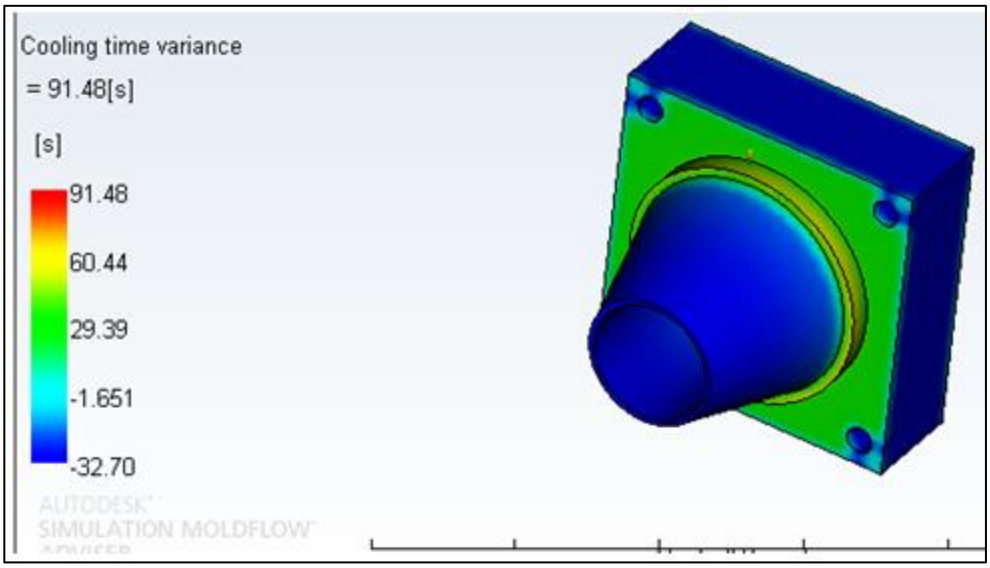

Fig 8: Cooling Time Variance Results of the Mold Flow Analysis

V. DISCUSSION

Figure 1 shows the gate location result for the mould. The best location for the injection gate is highlighted in blue colour whereas the location that is most unfavorable is highlighted with red colour. Simulation was carried out to predict the best and worst location for the gate placement. This simulation helped in providing important data to the designer about the placement of gate and then the feeding system can be designed according to the placement of gate.

As the cavity fills, Figure 2 displays the flow front's fill time result location at regular intervals. The contour colors that depict the plastic flow into the part are displayed in the diagram. At the same time, every region with the same color is filled. At the beginning of the filling (injection), the outcome was seen to be dark blue, and the final sections to be filled were red. Using filling times of 0.000 seconds, 2.112 seconds, 4.224 seconds, 6.336 seconds, and 8.448 seconds, the results showed that the flow pattern is balanced in a section with a good fill time result. The area that did not fill will be colorless if the portion was short shot (a short shot will appear translucent). All flow paths completed simultaneously and reached the model's edges; b) the contours were uniformly spaced and showed the speed at which the polymer was flowing; c) the contours were widely spaced, indicating rapid flow; and d) the contours were narrow, indicating slow filling. These characteristics indicate the uniformity of filling.

The simulation results in terms of time to reach part eje ction temperature (time to freeze) indicate that normal cooli ng channels took around 75 seconds to reach ejection temper ature for most of the plastic parts, as indicated by the blue co lors contour in figure 4.3. The time required to reach the eje ction temperature was measured from the start of fill and too k into account the dynamics of the packing phase and where new hot material enters the cavity, which affected the coolin g time. It was observed that the part frozen uniformly, which is an ideal situation to the time to reach ejection.

The fill analysis's injection pressure result, shown in Fi gure 4, indicated that a maximum injection pressure of 1.787 MPa was needed to fill the cavity.The pressure was zero on the absolute pressure scale at the start of filling.At a particul ar point, the pressure begins to rise from 0.4469 MPa, which

ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

has a dark blue contour, to 0.8937 MPa, which has a light bl ue contour.Because of the growing flow length between this particular spot and the melt front, the pressure keeps rising a s the melt front moves.This is followed by a green contour at 0.8937 MPa, a yellow contour at 1.341 MPa, and a red cont our at 1.787 MPa.During filling, the force pushing the poly mer melt to flow is the pressure differential between locatio ns.The polymer always flows in the direction of the negative pressure gradient, from higher pressure to lower pressure, m uch like water does from higher elevations to lower elevatio ns. As a result, during the filling stage, the melt front experie nces the lowest pressure and the polymer injection locations the highest pressure, as illustrated in figure 4.

The volumetric shrinkage upon ejection, represented as a percentage of the initial modeled volume, is displayed for each area in Figure 5.The drop in local volume from the conclusion of the cooling phase to the point at which the part has cooled to the ambient reference temperature—by default, 25°C—is known as volumetric shrinkage at ejection.

Additionally, the volumetric shrinkage data was utilize d to determine whether the model had sink marks.Warpage was reduced since volumetric shrinkage was found to be con sistent over the whole portion, with shrinkage values of 0.36 84%, 4.238%, 8.108%, 11.98%, and 15.85%.This is crucial f or the part's structural and aesthetic integrity as well as for th e material's proper packing.

Based on the findings of the shrinkage analysis, It can be infer that:

- The linear shrinkage of materials that shrink isotropically is around one-third of the volumetric shrinkage.
- The effects of orientation and relaxation determine the li near shrinkages in thickness, flow, and transverse directi ons for molded materials.

The sink marks in figure 6 have been reduced from 0.3 114mm with a red color contour to 0.000mm with a blue col or contour, which is appropriate for a typical cooling channe l.This outcome demonstrated a wellmanaged increase in pac king pressure and duration.Thus, it can be said that the beha vior of sink marks is dependent on the volumetric shrinkage value.The presence and location of sink mark or voids is det ected by the appearance of marks on the surface.This conclu sion also corresponded to Omar et al (2018) that Sink marki ngs occur at places with considerable local shrinkage.Sink m arks frequently develop in moldings with thicker portions.It also occurs as a result of the potential for uneven heat loss d uring cooling.

It was observed that the temperature of the polymer began flowing at 111 OC with a blue contour color, progressed to 145.8 OC with a light blue contour color, and continued to increase in temperature from 180.5 OC with a green contour, 215.3 OC with a yellow contour, and until the flow front reached a specified point in the center of the plastic cross-section of 250.00 OC with a red contour color. Figure 7 indicates the temperature at the flow front result. The material temperature at each position as it was filled is represented by the color contours. The outcome demonstrates how the flow front's temperature fluctuates during filling. During the filling phase, the flow front temperature shouldn't decrease by more than 2°C to 5°C.Hesitancy could lead to a short shot if the flow front temperature is too low in a thin section

Ofthe part.Surface imperfections and material deteriora tion may arise in regions where the flow front temperature ri ses by a few degrees.

The Cooling time variance result shown in figure 4.8, shows the difference between the time it takes for polymer to freeze in any region of the part and the average time it takes to freeze in the entire part. The average time to freeze is calculated on the basis of the surface area and the existing cooling channels. Areas that are plotted as positive values, which appear in red, take longer to freeze than the average time to freeze. Areas that are plotted as negative values, which appear in blue, freeze more quickly than the average time to freeze. Zero values in this result indicate the average time to freeze. Red areas demonstrate higher temperatures than the average part temperature, a sign indicating that the area will need more cooling. Taken together, the Temperature variance and Cooling time variance results indicate places on the part that might require redesigning, such as changing the thickness of a wall or the inclusion of cooling devices in the mold (such as a bubbler), or positioning cooling away from fast cooling areas. The Cooling time shows that there was variance in the result which is based on the time required for every part of the model to freeze completely. Using this result, you can decide whether to extend the estimated cycle time so that critical areas of the part are properly solidified before ejection.

VI. CONCLUSION

Using Autodesk Mold Flow Advisor, a mold flow anal ysis was performed on the toilet brush cup mold.The optimal gate location, fill time, time to attain the mold's ejection tem perature, injection pressure, volumetric shrinkage at ejection , sink mark estimate, flow front temperature, and cooling tim e variation, among other outcomes, were all satisfactory, as s een by the results above.

The findings showed that minimal molding flaws migh t be produced in injection-molded components.

REFERENCES

[1]. Jagannatha Rao M B, Dr. Ramni (2013); Analysis of Plastic Flow in Two Plate Multi Cavity Injection Mold for Plastic Component for Pump Seal published in International Journal of Scientific and Research Publications, Volume 3, Issue 8, August 1ISSN 2250-3153 www.ijsrp.org

[2]. Effects of injection molding conditions on the mechanical properties of polyamide / glass fiber composites by prof. Dr. lk YILMAZER Supervisor.

ISSN No:-2456-2165 https://doi.org/10.5281/zenodo.14862896

[3]. Adekunle A. Fagade and David O. Kazmer, Early cost estimation for injection molded parts, a Thesis by University of Massachusetts
[4]. J.A. Brydson, (2015); Plastic materials British Cataloging sixth edition, , Butterworth Heinemann Ltd.
[5]. Haddout,A & Villoutreix, G. (2017); An Experimental study of fiber orientation in injection molded short glass fiber reinforced polypropylene /polyacrylamide composites (Conservatoire National des Artset M6tiers, France), pp.147-153,