International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Optimization of Gating System Parameters for 20 Liter Water Jar Handle Component by Simulation Flow Analysis using NX-Easy Fill Advance Software

Ramu M. S.[1], L. G. Sannamani[2], Dr. S. N. Ravi Shankar [3]

[1]*Student, M.Tech in Tool Engineering, GT&TC, Mysuru-16, Karnataka, INDIA*
[2]*Deputy General Manager, GT&TC, Mysuru-16, Karnataka, INDIA*
[3]*Principal, Post Graduate Studies, GT&TC, Mysuru-16, Karnataka, INDIA*

ABSTRACT

Gate connects the runner to the mould cavity, making it an essential flow path in the gating system i.e. the sprue, runners and gates. The channels, through which the polymer flows, make up the injection moulding feed system [1]. Cycle time optimization, scrap avoidance and manual interface play a crucial role in enhancing process productivity while ensuring that the quality of the finished product is not compromised in the manufacture of plastic parts. This study employs NX-Easy Fill Advance software to examine the effects of the optimal gate placement on filling time and defects over a number of iterations. The optimized parameter was obtained using NX-Easy Fill software and used to designing the required moulds.

Keywords: Plastic Injection Mould, NX Easy- Fill Advance, Gate System.

1. INTRODUCTION

One of the most popular processes for manufacturing products made of plastic is injection moulding [2], which finds applications in many industrial as well as household consumer products [3].To make the right mould, you need An effective moulding tool. The present work is the design of the 20-liter water jar handle. While perform mould flow analysis, a 3D model is created with NX-11.0 software. The model is meshed, a suitable material is assigned and the best gate location was determined. All this steps were followed in the first phase. The processing criteria were established using the conventional trial-and-error method, which was frequently inadequate and impractical for intricate parts. As a result, in order to achieve the best outcome, iterations with increasing injection positions were carried out. One of the outputs was found to be satisfactory for all parameters, encouraging oneself to design and manufacture the mould.

2. OBJECTIVES

- Component research and development.
- Design and analysis of mould [4].
- Injection mould calculation and conceptual design.
- To manufacture a component that is free of defects

3. METHODOLOGY

Specific planned procedures or techniques used (identify, select, process, and analyze information) to design an injection mould to achieve the satisfactory results is called methodology. Fig 1 shows Procedural steps to design a mould tool.

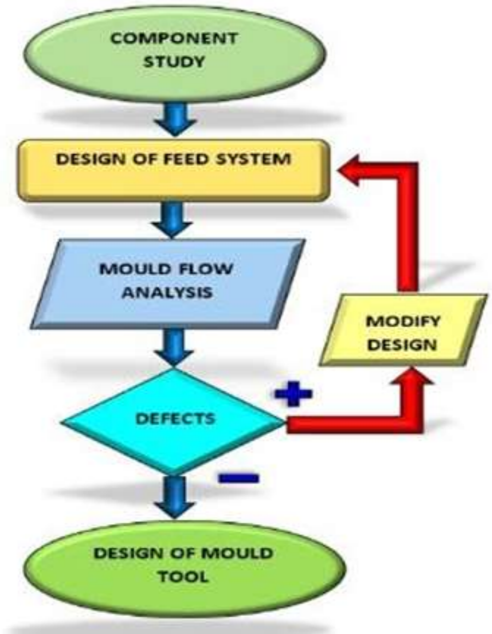

Fig. 1 – flow chart of methodology.

3.1 Component study

Component study is the major step to design a working tool successively. From 3D and 2D model, maximum data can be extracted easily. From this, points to be studied are as follows,

- Geometry of the component such as critical dimensions, thickness and other dimensions can be understood easily through 3D & 2D models.
- Type of material used to design a component and check whether the material is suitable or not by their properties.
- The suitable quality of material checked to produce the component as well as mould tool.
- Factors to design mould are parting surface, undercuts, drafts, core & cavity. Achievable aspects of the component in the mould for easy ejection are some of the details fixed from the component study.

Polypropylene (PP) 98% is used as material for the component. The component is green in color. To make the part of the desired color, 2% green master batch is used. The component's volume is 18.4 cm^3. The total weight of the components is 16.24 g. The component has a density of 0.898 g/cc. The melting point is195 °C. The component 3D model is as shown in Fig.2.

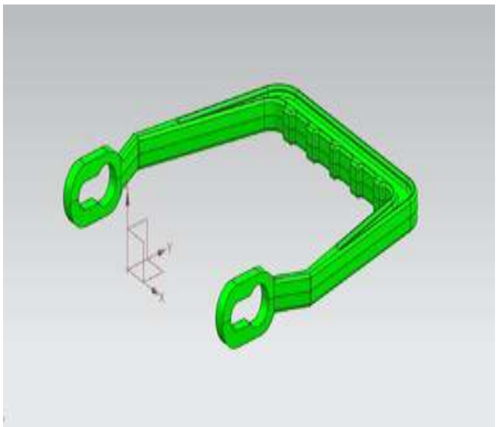

Fig. 2 - 3D model of jar handle.

3.2 Design of the Mould Tool

Calculations are made to choose the suitable machine to run the production of mould.

Major calculations are as follows,

- Determining the shot weight of the component as well as the machine.
- Plasticizing capacity of the machine at required material selected.
- Clamping tonnage required for the component.
- Injection pressure required to fill the mould completely.
- From the above calculation, it is possible to select the parting line/surface, number of cavities, feed system, thickness of the plates, inserts, finally cooling line and cooling location.

3.2.1 Runner design [6]

$$\text{Diameter of runner} = \frac{\sqrt[4]{L} \times \sqrt{W}}{3.7} \qquad \ldots (1)$$

Where,

W= weight of the component = 16.469 g

L = length of the runner = 110 mm

$$\text{Diameter of runner } (\emptyset) = \frac{\sqrt{(4\&220)} \times \sqrt{36.23}}{3.7}$$

Diameter of runner (∅) = 6. 26 mm

For ease of machining and ease of cutter availability, it is rounded off to 6 mm. (because of increase in quantity of scrap and limited cutter availability)

3.2.2 Sprue design [5]

A standard sprue should adjust to the diameter of the runner with one end of it and the other end's diameter is analyzed using the relationship given below,

$$\mathbf{D_1 = D_2 + 2(SH)\ tan\ A} \quad \ldots.. (2)$$

Where,

D1 = Sprue diameter at the upper end = diameter of runner = 6 mm,

D2 = Sprue diameter at the lower end

SH = Length of sprue chosen by the designer = 60 mm and

A = Tapered angle (20 - 50) = 20 taken

$$\mathbf{D_1 = 6 + 2(60)\ tan\ (2) = 10.19\ mm}$$

As a result, the sprue diameter is 10.19 mm, which is rounded off to 10 mm. (because of increase in quantity of scrap and limited cutter availability)

3.2.3 Gate design [5]

3.2.3.1 Gate width (Wg)

Width of the gate (Wg) is calculated by:

$$Wg = (n \times \sqrt{A})/30 \qquad ,,,,,, (3)$$

Where n is the material constant, and for PP n = 0.8

$$Wg = \frac{0.8 \times \sqrt{12167.94}}{30}$$

$$Wg = 2.94\ mm$$

3.2.3.2 Gate height (h)

$h = n \times t$ …... (4)

where, n = 0.8

t = Average wall thickness = 2.34 mm

h = 0.8 x 2.34

h = 1.872 mm

3.2.3.3 Gate length (L)

$$L = h + \frac{Wg}{2} \quad \text{…. (5)}$$

$$L = 1.9 + \frac{2.94}{2}$$

$$L = 3.34 \text{ mm}$$

Therefore, the gate's dimension is **3.34 x 2.94 x 1.87 mm.**

Fig. 3 and Fig.4 shows the design of feeding and gate system

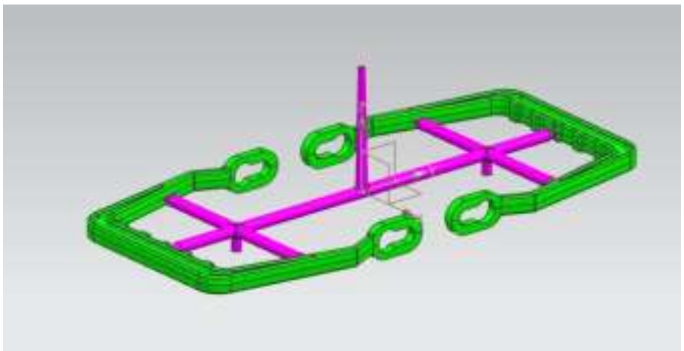

Fig. 3 - feeding system design.

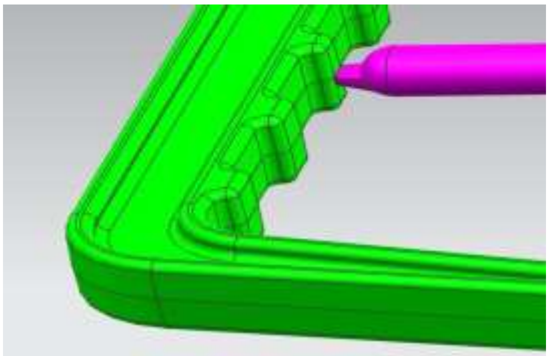

Fig. 4 - gate system design.

3.3 Mould flow analysis

Software such as Mould Flow Advisor, NX Easy Fill Advance etc. which gives the simulations and best results from relevant parameters are used. This software also helps to identify the defects in the component. From this, product quality can be analyzed [7]. Few results are given below,

- Type of gate and location of gate can be analyzed.
- Injection time or time of filling can be obtained.
- Confidence of fill, filling time, melt front temperature, quality prediction, pressure drop, clamping force, average temperature can be analyzed.
- Defects such as air-traps and weld-lines can be analyzed.

In order to make high-quality components, Mould flow analysis is performed; it is a study of flow of material that helps to evaluate the required parameters, mould design and component [8]. NX Easy-Fill Analysis is controlled by the Moldex3D application [9]. Early in the process of developing a product, designers can test the mouldability of plastic part designs using this integrated tool for mould flow simulation. Additionally, developers can make changes in advance to enhance gate number/locations [10], material selection, part design, process conditions, or material selection [11].

Table 1 shows Material used for component is polypropylene (PP) and It’s density, Melt Temperature, Temperature, Pressure.

Table 1- Input data for analysis

Material selected	PP
Supplier	Advanced composites
Trade name	ADX 1056
Density	0.898 g/cc
Melt Temperature	195 °C
Mould Temperature	54 °C

Max. Pressure	70 MPa

3.4 NX- Easy Fill Advance Results

The mould flow simulation is performed for several numbers of times for different positions and parameters. The best iterations chosen for manufacturing based on gate contribution, fill time, pressure and air traps as shown in Table 2.

Table 2 shows the fill time is the amount of time it takes the polymer to entirely fill the core and cavity. Fill time 2.4 s at a pressure of 10.87 MPa and a temperature of 195 °C. Air traps and weld lines are Acceptable.

Table 2-Results

SL. NO.	PARAMETERS	RESULTS
1	Gate contribution	Balanced flow
2	Melt front time	2.4 s
3	Pressure	10.87 MPa
4	Temperature	195 °C
5	Air traps	Acceptable
6	Weld lines	Acceptable

3.4.1 Gate Contribution

The design and simulation of the feeding system in the six gate system uses six gates with a minimum thickness and runner diameter of 6 mm. The melt contribution to each gate in a six-gate system is balanced.

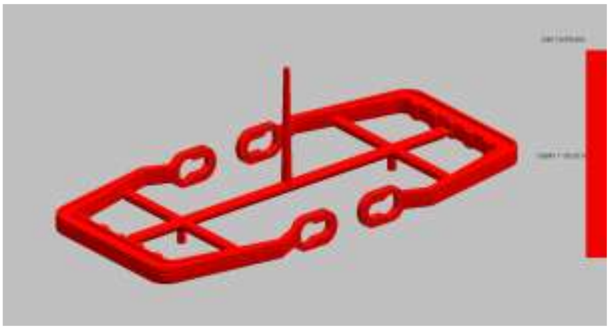

Fig5 - gate contribution.

3.4.2 Front Melt Time

Front melt time is the amount of time it takes for the molten plastic to entirely fill or flow into the cavity. The cavity will need to be filled in 2.4 s. The production rate rises as filling the cavity takes less time. Better gate systems are preferred because they fill the cavity in an even, balanced manner in less time and also producing goods of the highest quality, which helps prevent flaws.

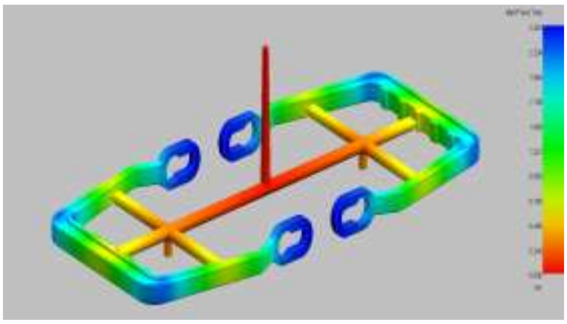

Fig. 6 - melt front time

3.4.3 Pressure

Injection moulding, pressure is a crucial parameter. Incorrect fill pressure can cause defect like mould flashing, which degrades the component's surface finish. The fill pressure (10.87 MPa) should therefore not to be close to the machine pressure in order to achieve better surface quality.

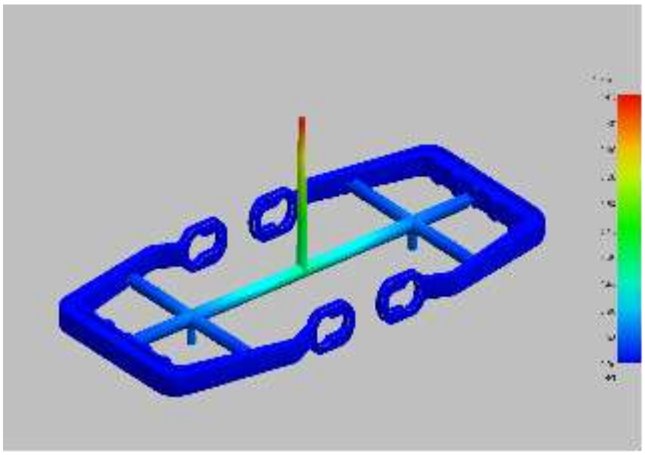

Fig. 7 - pressure.

3.4.4 Temperature

Temperature (195 °C) increases cause flaws like surface cracks and weld lines, which lowers the quality of the finished goods.

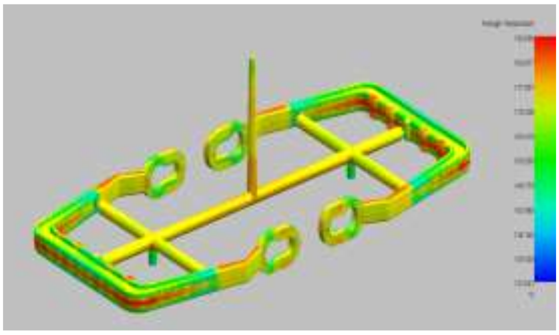

Fig. 8 - temperature.

3.4.5 Air traps

There is air traps located within the hollow. This might cause cavities, bubbles, or a fault in the component's surface. Air traps can be reduced by incorporating air vents in the core and cavity.

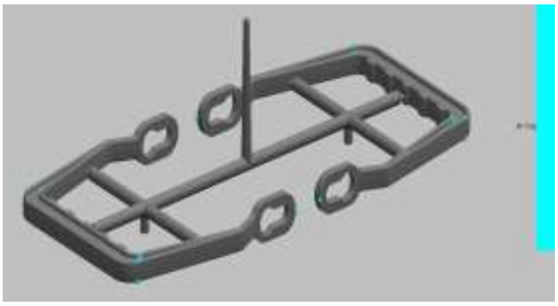

Fig.9 - air traps.

3.4.6 Weld lines

Weld lines are one of the defects examined in the injection moulding process; while they cannot be completely eliminated, they may be reduced by using specific design factors that the designer should take into account. The strength and appearance of the item are adversely affected by weld lines. The temperature of the mould will cause any established weld lines to melt, improving the surface smoothness. By monitoring the injection pressure, barrel speed, and mould temperature, the weld line may be adjusted.

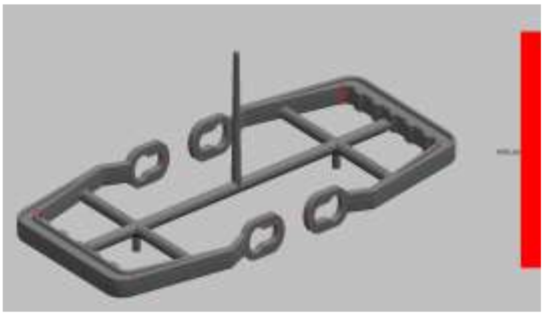

Fig.10 - weld lines.

CONCLUSIONS

1. A proper flow chart of design of mould for present study using 3D and 2D model.
2. The component's name is 20 liter water jar handle, 98% of Polypropylene (PP) material along with green color of 2% used per batch.
3. The component volume is 18.4 cm3 and the total weight is 16.24 g and density of 0.898 g/cc with melting temperature of 195 °C.
4. Designing of mould performed using runner, sprue, and gate design. Obtaining runner diameter is 6 mm, sprue diameter at lower end is 10 mm, gate width is 2.24 mm, gate height is 1.872 mm and gate length is 3.34 mm.
5. NX Easy Fill Advance Software is used to obtain the optimized parameter for above study. Obtain Results as mentioned in table 2.
6. The above article is used to perform in a planned manner to design required mould.

References

[1]. Vishnu Darshan M. (1*) , Mr. Govind Vatsa(2),Dr. S. N. Ravi Shankar(3) “Influence Of Gating System For Top Cover Strut Bearing Component By Simulation Flow Analysisusing NX-Easy Fill Advanced Software.” Published:2021.

[2]. P D Kale(1*), P D Darade(2) and A R Sahu(1), “A literature review on injection moulding process based on runner system and process variables”, Published: 2021.

[3]. Prashant K Mulge(1), Dr. Shashidhar S Kalashetty(2),“A Brief Literature Review on Optimization of Plastic Injection Moulding Process Parameters for Various Plastic Materials by using Taguchi’s Technique”, Vol. 8 Issue 07, July-2019.

[4]. Vijaykumar Vilas Andhalkar (1) , Dr. S. R. Dulange(2),”Injection Molding Methods Design, Optimization, Simulation Of Plastic Flow Reducer Part By Mold Flow Analysis.” Volume: 04 Issue: 06 | June -2017.

[5]. Kiran Tom Thomas(1*) and Ramesh Babu K(1), “Conceptual Design of Two Plate Injection Mould Tool for Five Pin Daimler Regulator” ISSN 2278 – 0149 ,www.ijmerr.com,Vol. 3, No. 4, October 2014.

[6]. Lokeswar Patnaik(1), Sunil Kumar(2), “Analysis And Design Of Multi Impression Split Core, Finger Cam Operated Mold For Brass Insert Connector Plug”, April 2017.

[7]. Manmit Salunke(1), Rushikesh Kate(2),Vishwas Lomate(3), Gajanan Sopal,” Injection Molding Methods Design, Optimization, Simulation of Plastic Toy Building Block by Mold Fllow analysis.” Volume 6, Issue 6, June (2015).

[8]. Hillary Ejike Chukwu(1*), Harold Chukwuemeka Godwin(2), Uchenna Samuel Ugwu(1), “Computer Aided Design and Simulation of Bottled Water Handle”, Received: September 13, 2016; Accepted: October 1, 2016; Published: October 26, 2016.

[9]. Viswa Mohan Pedagopu(1) & Manish Kumar(2) ,“Design And Machining Simulation Of a Prismatic Part Using Nx Cad/Cam An Overview”, February 2014.

[10]. Wong(1), C. T., Shamsuddin Sulaiman(2), Napsiah Ismail & A.M.S. Hamoud(3), “Design and Simulation of Plastic Injection Moulding Process”, January 2004.

[11]. Vasco, J.(1), Capela, C.(1),ártolo, P.(1), Granja, D.(2), “Material Selection For High Performance Moulds.” November 2007.