IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

MOLD FLOW ANALYSIS FOR OPTIMISATION OF GATE LOCATION IN AN INJECTION MOULDING TOOL FOR AN ELECTRIC METER BOX.

[1]Goutham P K, [2]Dr. N Ramesha

[1]Student, MTech in Tool Engineering, GTTC, Bengaluru, India
[2]DGM, Training Department, GTTC, Bengaluru, India

Abstract: The gate connects the runner to the cavity, making it an essential flow path in the feed system. Sprue, runner systems, and gates, the channels through which the polymer flows, make up the injection moulding feed system. 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 solidworks software over a number of iterations to examine the effects of the optimal gate placement on melt front time and defects. The ideal gate placement is chosen following the iterations.

Index Terms - Plastic Injection Mould, Solidworks, Feeding unit,* Acrylonitrile Butadiene Styrene (ABS), *Pinpoint gate, Gate contribution, Defects.

1 Introduction

Due to their many useful qualities, plastic products are very popular right now. One of the most popular processes for creating plastic products is injection moulding. To make the right mould, you need a moulding tool [1]. The moulding of the electric metre box is the main topic of this study. In this instance, a 3D model was created using the software Solidworks, and mould flow analysis was then performed. The model was meshed, a material was assigned, and the best gate location was determined. All of the steps were finished in the first phase. The processing criteria are established using the conventional trial-and-error method, which is usually inadequate and impractical for complicated parts. In order to achieve the optimal result, two iterations with increasing injection positions were carried out. One of the outputs was declared satisfactory for all parameters, encouraging the designer to move forward with the design and manufacturing process and procedures.

2 Objectives

- Component research and development.
- Injection mold calculations and conceptual designing.
- To manufacture the component which is free of fundamental flaws.

3 METHODOLOGY

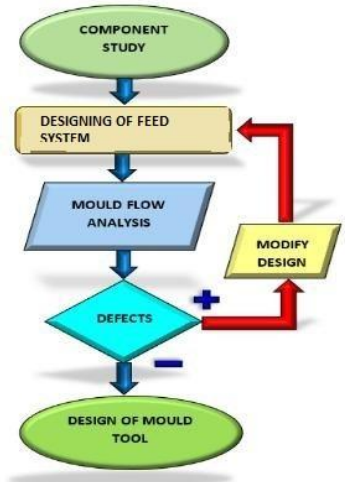

Figure 1 Flow chart of Methodology

3.1 Component study

Electric meter box is the name of the component. Component material is made up of ABS. Dimension of the component is 95x95x60mm. Volume of the component is 62430.08 mm^3. The component weight is 63.68 grams. Density of the component is 1.01 grams per cubic centi-meter and 0.5 percent of the original volume has shrunk. Melting temperature is 200 ℃ .

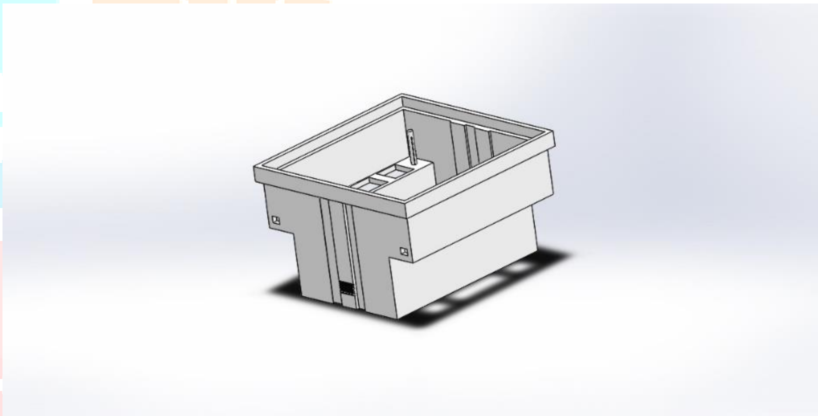

Figure 2 Electric meter box

Table 1 Component data

PROPERTIES	UNITS	SYMBOLS	VALUES
Density	g/cm^3	ρ	1.01
Volume of the component	cm^3	V	62.43
Shrinkage	%	s	0.5
Injection pressure	bar	p	875
Average wall thickness of component	mm	δ	2.5
Projected area	cm^2	PA	570
Moulding temperature	℃	T_{mould}	80
Melting temperature	℃	T_{melt}	200

3.2 Design of Feed system

- **Runner design**

$$Diameter\ of\ runner = \sqrt{W} x\ \sqrt[4]{L}\ /3.7 \quad \text{Eq1}$$

Where,
w = weight of the component in g = 67 g
Lr = length of the runner = 40mm
Diameter pf runner(∅) = ($\sqrt{67}$ x $\sqrt[4]{40}$)/3.7 = 5.6 ≅ 6mm
For ease of machining and ease of cutter availability it is rounded off to **6 mm.**

- **Sprue design**

$$D_1 = D_2 + 2L\tan A \quad \text{Eq2}$$

Where,
D_1 is the diameter of sprue at upper end,
D_2 is the diameter of sprue at lower end i.e., 6 mm end,
SH or L= length of sprue selected by the designer = 40 mm and A = tapered angle (2˚ to 5˚) =2˚ taken
$D_1 = D_2 + 2L\tan A = 6 + 2(40) \times \tan(2^\circ) = 8.75$
Hence the diameter of sprue is 8.75 mm which is rounded off to **9 mm**

- **Gate design**

$$D = n \times c \times A^{1/2} \quad \text{Eq3}$$

Where,
D = diameter of the pin point gate
A = projected area = 9025mm^2
n = Material constant = 0.6
c = Function of the wall section thickness (Appendix 3A) = 0.326mm
D = 0.6 x 0.326 x 9025$^{1/4}$
D = 1.5mm
L = length of the gate =1 to 3mm
Length of the gate is considered to be 3mm.

Sprue gate can be considered but due to its large degating impression it is not suitable for this component. Due to this a pin point gate is considered because this is used with a three plate injection mold tool, this feed system can be degated automatically and this will not leave a major gate mark on the component. Gate diameter of 1.5 mm and 3 mm of gate length is considered and a semicircular runner is considered as ABS is a medium viscous flow material [2-7].

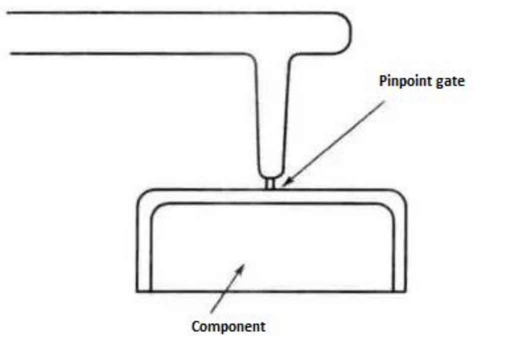

Figure 3 Pinpoint gate

3.3 Mould flow Analysis

In order to make high-quality parts, mould flow analysis is a study of the flow of plastic materials that helps evaluate the component, parameters, and mould design. Early in the product development process, designers can test the moldability of plastic part designs with solidworks flow analysis, an integrated mould flow simulation tool. Additionally, developers can make changes in advance to enhance gate number/locations, material selection, part design, process conditions, or material selection [8-9].

1st Iteration results

Fill time is the amount of time taken by the polymer to entirely fill the core and cavity. Because it affects how rapidly the polymer is injected into the mould, it is an important factor to consider. The portion fills in the first iteration, shown in Fig. 4, in 2.308 seconds at a pressure of 33.16 MPa. The surface quality of the component will be impacted by defects such mould flashing if the fill pressure is too high or too low. It is obvious from Fig. 4(b) that pressure increases as the cavity and core are filled. When air cannot escape through the vents or inserts provided in the mould, it becomes trapped in the blue zone or dots observed in Fig.4(c). The material utilised, as well as the component's design and structure, are the main causes of weld lines. When polymer flows from opposing directions collide in the same region, they occur. Figure 4(d)'s red lines show the creation of weld lines in the component.

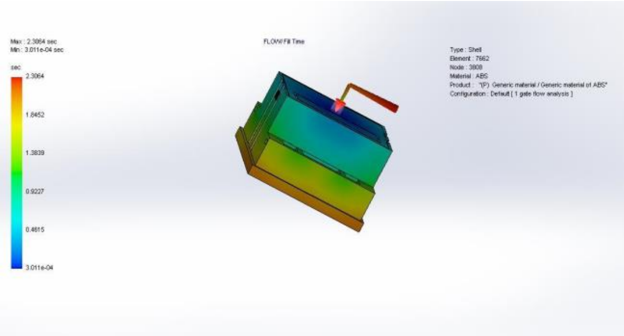

Figure 4(a) Fill time

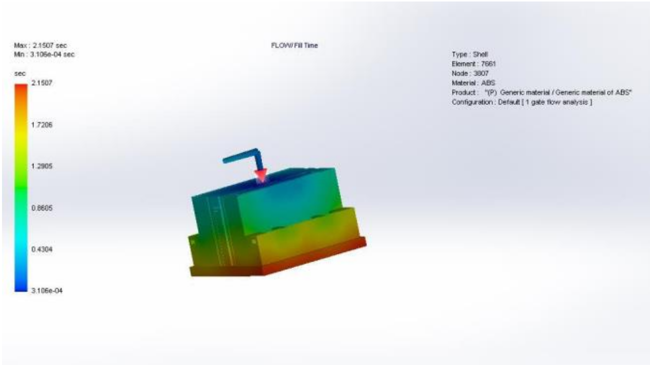

Figure 4(b) Pressure

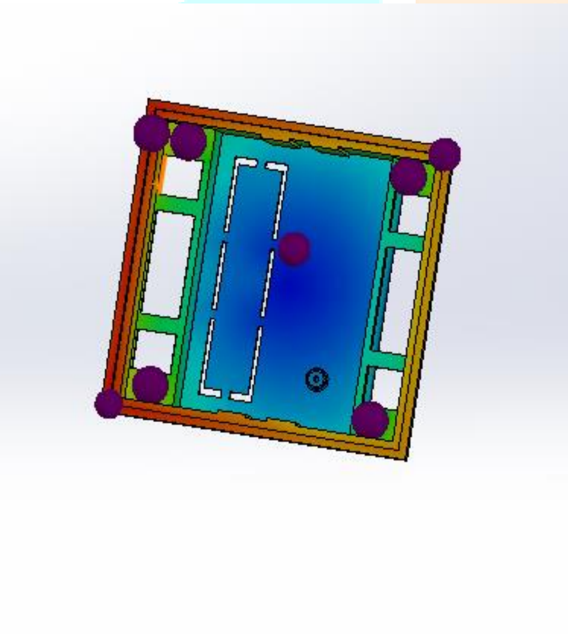

Figure 4(c) Air traps

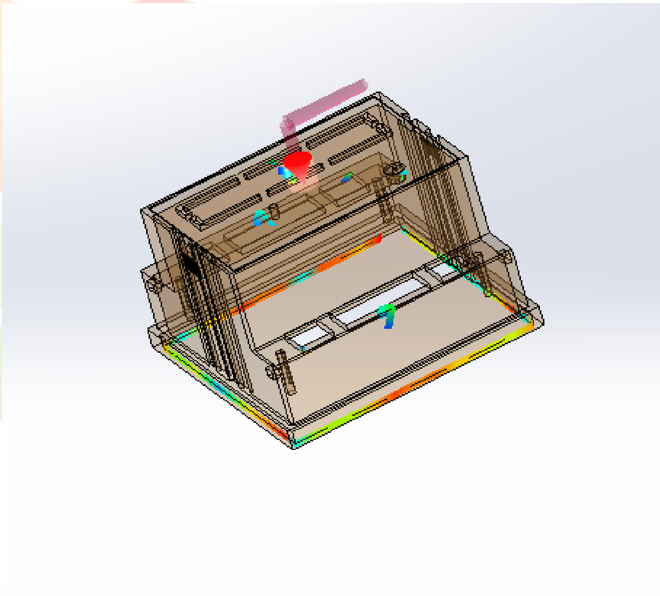

Figure 4(d) Weld lines

Table 2 1st iteration results

SL.NO.	PARAMETERS	RESULTS
1	Melt front time	2.308 s
2	Pressure	33.16 MPa
3	Temperature	230°C
4	Air traps	Minimum
5	Weld lines	Minimum

2nd iteration

The results from the first trail can be further adjusted by adjusting the number of gates, here two gates are used with runner length of 60mm. Where the component fills in 2.115 seconds, which is better than previous result with a pressure of 32 MPa. The pressure is decreased compared to previous results. Air traps, weld lines were acceptable. This result leads to good surface finish of the component. Hence 2 gates system has been approved for further manufacturing process.

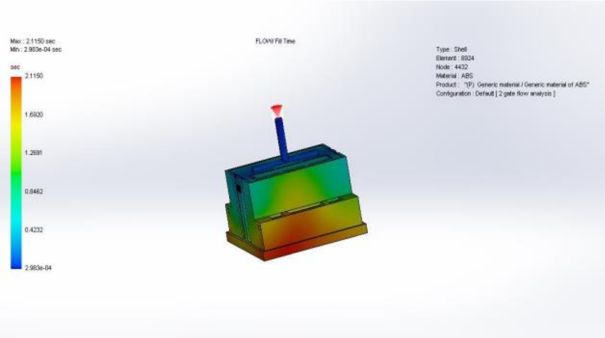

Figure 5(a) Fill time

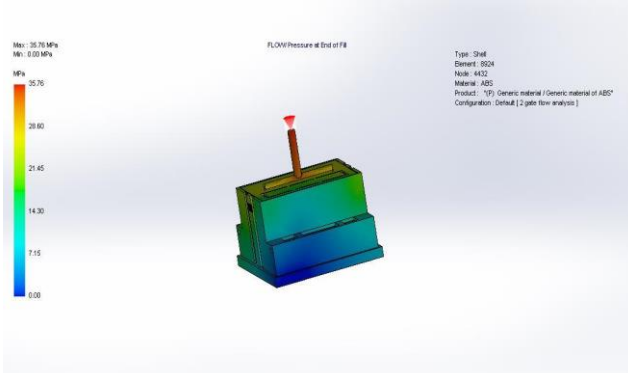

Figure 5(b) Pressure

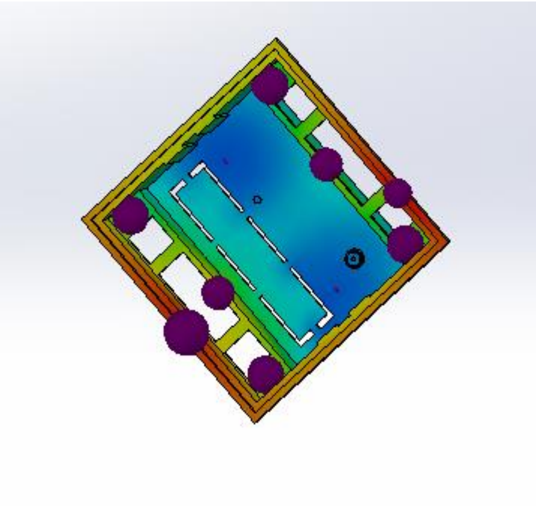

Figure 5(c) Air traps

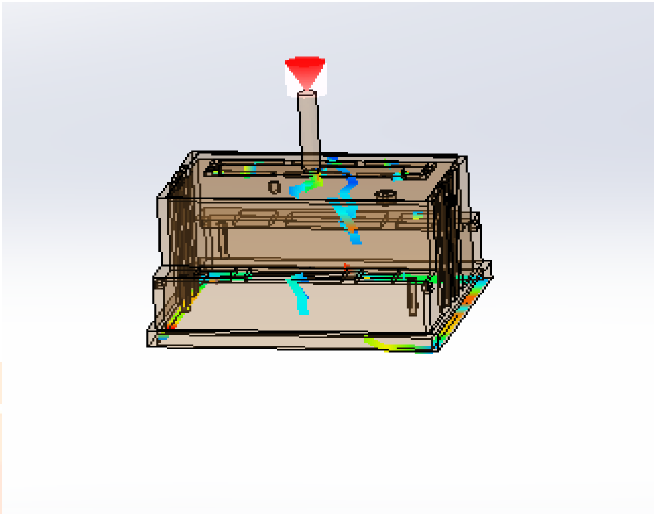

Figure 5(d) Weld lines

Table 3 2nd iteration results

SL.NO.	PARAMETERS	RESULTS
1	Melt front time	2.115 s
2	Pressure	32 MPa
3	Temperature	230°C
4	Air traps	Acceptable
5	Weld lines	Acceptable

4 CONCLUSIONS

By contrasting the aforementioned two findings, the analysis for the best gate and runner placement was clarified, and errors in the flow simulation such air traps and weld lines were removed. Because the gate is positioned for easy production and defect-free components, automatic de-gating is possible. By incorporating air vents in the core and cavity inserts, air traps can be decreased. By monitoring injection pressure, barrel velocity, and mould temperature, weld lines can be controlled. From Table4, 2 gate points have been accepted for the next step in the production process.

Table 4 Comparison of two iteration results

Sl. No.	PARAMETERS	1 GATE POINT	2 GATE POINTS
1	Moldability	Good	Better
2	Melt front time	2.308 s	2.115 s
3	Pressure	33.16 MPa	32 MPa
4	Temperature flow	230°C	230 °C
5	Air traps	Minimum	Acceptable
6	Weld lines	Less	Acceptable

5 REFERENCES

1. Plastic Injection Moulding Machine", Gaikwad Mahesh Digambar, Gaikwad Mahesh Digamber1, Sharif Mohiuddin Akbar2, Dhomase Datta Govind3, Ravale, Shivshankar Ashokrao 4, Mr Bidve Mangesh Angadrao 5, Mr Devshette Ashish Rajkumar 6, Volume: 06 Issue: 03 | Mar 2019.
2. "Studies on Acrylonitrile Butadiene Styrene (ABS) and Electroplating on ABS" Jayanth.K.N et al.,
3. "Injection Mould Tool Design of Power Box Side Panel" B.Iftekhar Hussain et al.,
4. Injection mould design, R.G.W Pye, 4th Edition, Affiliated East-West Press Pvt Ltd.
5. "Influence of gating system for top cover strut bearing component by simulation flow analysisusing nx-easy fill advanced software" Vishnu Darshan M et al.,
6. "Influence of feed system in Injection Moulding for Lower Washer of a Bearing", Vikas B J 1, Chandra Kumar R 2, Volume: 02 Issue: 08 | Aug-2013.
7. "Design of pin-point gate injection mould for covers of earplugs". Yu SR et al. 2016
8. " Mould Flow Analysis of a Plastic Injection Moulding Component Using Software Tool to Obtain a Quality Output Sowrabh Kugunavar et al.,
9. "Simulation for Optimum Gate Location In Plastic Injection Moulding for Spanner" Atul B. Munankar et al. (2019)
10. "Computer simulation opportunity in plastic injection mold development for automotive part." In Computer Technology and Development Saman et .al. 2009.