E-ISSN 2581 – 7957
P-ISSN 2277 – 3916
CVR Journal of Science and Technology, Volume 23, December 2022
DOI: 10.32377/cvrjst2321

Analysis of a Feasible Gate Location in Injection Mold for Plastic Cloth Peg: New Product Development

Neeraj Kumar Jha
Assoc. Professor, CVR College of Engineering/ Mechanical Engg. Department, Hyderabad, India
Email: neerajjha.me@gmail.com

Abstract: **Product design is the most important and invaluable factor to determine success of a product. Its role changes consistently throughout the life cycle of a product. It is well known that the ultimate design of a product is dependent on various influencing factors right from material and process selection to the end use as well as re-use of the product. These factors put challenges in front of designers and manufacturers of consumer goods. Judicious utilization of material and other resources along with optimized profit are the constraints to control. It is achievable only if the design concepts and the manufacturing experience is used in the correct combination. This article aims at illustrating a new product development process for a plastic part and analyzing it on the parameters of few process constraints related to its manufacturing process viz. gate location. The product considered for the study is clothe peg, which is individually made up of plastic, unlike existing similar products made in combination of plastic and metal parts. This part is analyzed by considering its manufacturing by injection molding process. A model mold was also prepared for validation of the research findings.**

Index Terms: **product design, injection molding, gate location, distortion, parting line, shrinkage.**

I. Introduction

Plastic is an engineering material with multiple engineering applications and advantages. The reusable grades of plastics become soft when they are heated. By applying pressure, they change shape at this softness to come to a usable form. This is the basic principle behind the processing of plastic. Injection molding process is one of the processes of processing thermoplastics with which 80% of the plastic products are manufactured. The process utilizes a plastic injection machine along with a permanent mold. The machine plasticizes the polymer granules by heating them and pressurizes them to fill the mold. The mold consists of cavity as the shape of the required product along with flow path to fill the cavity. The mold additionally consists of provision of cooling the injected polymer in its cavities. These cooling channels help in fast solidification and improve cycle time of the process. The process seems simple and controlled, but uncontrolled process parameters lead to defects in part produced, wastage of material, cost, and time in injection molding process. injection pressure, injection temperature, mold temperature etc. along with the processing material properties, are some process parameters which must be analyzed properly to ensure defect free components obtained by injection molding process. Not just process parameters but process planning also plays a great role in successful production of a product. At macro level process planning controls resource utilization to increase profit margin but micro level process planning controls all the steps with which overall efficiency of production can be maintained. One such tool is the injection molding die (mold) which is used in the injection molding process. Design of this mold is one of the crucial parts of the entire production process. A properly designed mold will be easy to manufacture and easy to operate. It will have least possible inserts or other parts so that it can last long with negligible maintenance. We must understand that such molds are made to improve the production efficiency and productivity while their cost must be kept as low as possible. Design of such mold will be based on the part which it has to shape, and its observation done by the mold designer. A designer must analyze the part, to make a durable mold at optimum cost and can shape defect-free products. Analyzing the product and modifying it for ease of its manufacturing is part of product design. Providing adequate fillet on corners and draft on vertical faces are few attributes of the product design process for injection molding. Out of many design constraints in a mold design, the gate location has its own importance. Gate is the last element of the flow path of a mold. It connects runner with mold cavity and through the gate only material enters the mold cavity. They are with the least cross-section compared to other elements of the flow path so that back flow of material can be prevented, and directional solidification can be ensured. There are many types of gates in mold making practice. They are used as per specific need. Edge gate with circular cross-section is more frequently used for ease of cutting. They leave a mark on the product. Thus, their position and type are considered specifically as per the use of the product.

The product considered in this work is clothes peg. It is not the same conventional clothe peg used for gripping the clothes on a rope while the clothes are getting dried. It is a new product to serve the same purpose, but it is different in shape, elements and appearance compared with the conventional cloth pegs. The existing pegs are mainly made of plastic, and they implement metal spring to serve their purpose. Usually, they don’t last long due to non-parallel characteristics of metal parts and plastic parts. This work aims at making the entire peg from a single piece of plastic. This work innovatively utilizes shrinkage occurring in a plastic part blended with shape transformation to replace metal spring from a plastic clothe peg. The design of the proposed clothe peg is influenced with the existing clothes peg but it is new of its kind. It is considered as new product development because the steps involved for a new product development

Received on 03-11-2022, Revised on 09-11-2022, Accepted on 17-11-2022.

process are followed [1]. From past research it is evident that there are seven most shouted steps in new product development. Many researchers utilize twelve steps of new product development. Thus, it is evident that as per the product complicacy and application, few of the stages can be obsolete or combined.

The article is structured in the following sections: Section II explains the product design process by considering stages of the new product development process in background. Section III explains process simulation by considering material and process parameter selection. Section IV is about die trials and Section V is the conclusion of this research work.

II. Product Design

There are seven main steps in the new product development process. They are, idea discovery, idea screening, content development and test, business analysis, development of mix between product and marketing, market test and product launch. This work discovered the idea based on the drawback of existing clothe peg. Over a period of use, plastic degrades while the metal spring remains stronger in the existing clothes pegs. Further use of such pegs leads to fracture of plastic part against metal spring and the part can no longer be used. Thus, there was a need for a durable peg. The idea screening is important and challenging at the same time. In this step there is a need to analyze the need or problem statement deeply. Here it is observable that durability issues are coming into picture due to incompatible combinations between metal and plastic parts. Metal spring was needed for stiffness and plastic part is needed due to its own advantages. In order to enhance durability, thicker plastic parts cannot be used as that will affect the cost of the product. Thus, durability can be aimed by plastic part alone which will limit product cost also [2]. But obtaining stiffness like a metal spring was another hurdle. Under content development, this work decides to utilize camber curve profile as solution for stiffness in plastic parts. The designed peg is shown in Fig. 1. Once molded as in figure, it will shrink in negative camber form. The gap between elongated parts will compress the cloth and rope together. Circular recess in the camber region is for receiving the diameter of hanging wire or rope.

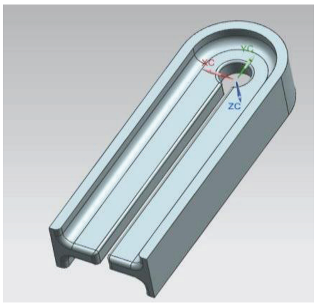

Figure 1. New Cloth Peg Solid Model

The finalized cross-section was T-section. Apart from these other feasible sections can be I and trapezoidal. Solid trapezoidal section cannot be a good choice as it will lead to a lot of shrinkage and thereby distortion of the molded product. Symmetric I-section cannot serve the purpose of cambering due to its symmetry. Modified I-section can be considered as alternate of trapezoidal section. But as per the application of the product we need high tensile stress on the outer surface and high compressive stress at the inner surface. It can be managed by varying material accumulation on inner and outer surfaces. Modified I-section will decrease intensity of compressive stress on the inner surface. This can be justified from calculation by considering the camber region as a curved beam or by stress analysis using convenient software. Stress analysis for T-section is shown in Fig. 2.

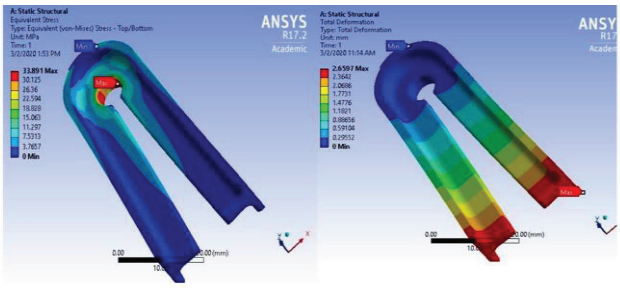

Figure 2. Equivalent Stress (L) and Total Deformation (R)

In Fig. 2, the left side image represents equivalent stress analysis while the right side image represents total deformation analysis performed by ANSYS R17.2. The maximum load implemented was 16.38 KN. This is the amount of load required to stretch a conventional clothe peg. This load value is obtained by experiment performed on a conventional clothe peg. It is observable from Fig. 2 that maximum stress was acting at circular recess and its magnitude is 33.89 MPa. Maximum deformation observed was nearly 3 mm at the free end. Both the values are in the agreement of design as it is not taking the part to ultimate failure against applied load and major volume of part material is receiving minimum stress as indicated in blue color. This can be considered as justification for another phase of product development process, content development and test. Hence, we can observe that the provided solution is an alternative to existing clothe pegs. It has an additional advantage for business analysis point of view over the conventional pegs. It can be observed that the existing pegs are made of minimum three number of parts ie. Two plastic parts and one metal spring. While the provided solution is constituted in a single piece. This adds direct benefit for business point of view. We know that for making more parts more time requirement will be there as well as to process plastic and metal parts separate machinery and equipment will be needed. Thus, it can be qualitatively estimated that the provided design solution can be developed in less investment as compared with the conventional pegs. Further, development of mix between product and marketing, market test and product launch are beyond the scope of this work as this work is dedicated to the initial phase of product development. The remaining phases are based on pushing the product in the market, getting feedback, and further analyzing the outcomes with different tools. Hence that broader scope of work is not included in this research work. The proposed peg was developed by considering its manufacturing by injection molding process, thus all the design considerations for the design was considered accordingly. Not just the strength criteria but application and appearance also were considered while finalizing the design. At the free end of the peg fillet is provided for the smooth engagement of peg against rope. Flat surfaces are mostly dominating and additionally curvatures are not added to keep machining cost of the mold on the lower side. The web and flange regions are given with different thickness to avoid thick mass accumulation and thereby distortion and shrinking during the molding process. The drawing of the proposed cloth peg is shown in Fig. 3.

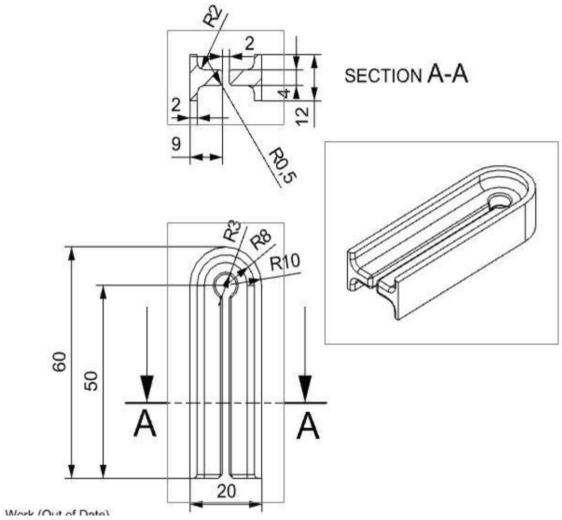

Figure 3. Drawing of New Cloth Peg (All dimensions are in mm)

III. PROCESS SIMULATION

This section will attempt to justify that the designed part is manufacturable. Suitable material selection, proper control on process parameters and process constraints will contribute to process simulation of the part. The same will be evaluated from the trial after manufacturing of the die.

A. Material Selection

The conventionally available cloth pegs are available in metal, wood and combination of plastic and metal. But this work aimed at making peg from plastic. There are a wide range of plastic materials suitable for the purpose, but this work proceeded with Polypropylene (PP) due to its own advantages.

TABLE I.
MATERIAL SPECIFICATION
POLYPROPYLENE (PP)

Density	946 kg/m^3
Melting Point	160 °C
Formula	$(C_3H_6)_n$
Type	Thermoplastic
Flexural Strength	40 N/mm^2
Shrinkage	1-2.5% mm/mm
Tensile Strength	32 N/mm^2
Injection Temperature	32-36 °C
Heat Deflection Temperature	100 °C
Specific Gravity	0.91

Few of the properties of PP are enlisted in Table I. Adequate flexural strength with low rate of shrinkage, compared with other thermoplastics makes PP suitable for this application. Additionally, PP has the least per unit cost among all polymers after polyethylene.

B. Process Parameter Selection

Selection of adequate process parameters is a very important phase of process design. Adequate injection pressure, shot volume, heating capacity of machine etc. are few important factors before starting the manufacturing activity [3]. As this work decided to make a model die for the trial of the part, a suitable machine was identified with adequate capacity for manufacturing of the part. Table II represents machine specifications of semi-automatic injection molding machines available for the trial.

TABLE II.
MACHINE SPECIFICATION
(TEXPLASST 1HD, MP LAB, CVRCE)

Shot Capacity	2 – 45 gms / shot
Plunger Diameter	25 mm
Stroke Lenght	450mm
Clamping Capacity	6.0 Tons
Injection Pressure	80 kg/cm^2
Heating Capacity	1.5 kw
Total Installed Power	3.7 kw
Total Shut Height	100 - 450mm

Apart from these process parameters few other aspects are also important to consider. One such aspect is the gate position of the part in the mold. By definition gate position is the part of flow path through which material enters into the mold cavity in the shaping process. But the decision of gating position is influenced by many factors. An inadequate gate position may lead to complicated and difficult to manufacture dies. Various defects may also appear in the components due to improper gating position. Gates are usually arranged along a parting line. But this decision mainly depends on the thickness of the part. The proposed part can have two convincing gate positions and the same is indicated in Fig. 4. Parting plane is indicated in red color in Fig. 4. It is passing through half of the thickness of the part. Probable gate position 1 (GP1) is indicated with a dark red arrow while probable gate position 2 (GP2) is indicated with purple color arrow. Both the positions have their own advantages and limitations. GP1 has the advantage that it is at an axis of symmetry, but die will be bigger while if GP2 is preferred, die size will be smaller and cost saving is possible but part quality is to be evaluated. This work further compared outcomes of each gate position through simulation.

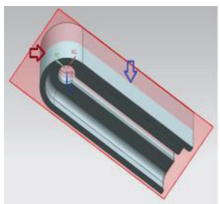

Figure 4. Probable Gate Positions and Parting Plane

E-ISSN 2581 – 7957 CVR Journal of Science and Technology, Volume 23, December 2022
P-ISSN 2277 – 3916 DOI: 10.32377/cvrjst2321

C. Simulation Results

Molded part quality is important to be observed before going for die manufacturing [4]. Thus, this work analyzes part quality for both the gate positions. Autodesk Mold Flow Adviser 2019 is utilized for this purpose. Such tools are conveniently utilized in manufacturing industries. It helps in a lot of time and cost saving in product design and development process. Fig. 5 represents confidence of filling the cavity, under common filling pressure. It can be seen that if filling is done from GP1, 100% high chance of filling is predicted while that of GP2 is 71%. The reason behind this result is that from GP1 feeding is uniform in both the arms while filling from GP2 one arm is getting filled easily and another arm fills later by residual pressure.

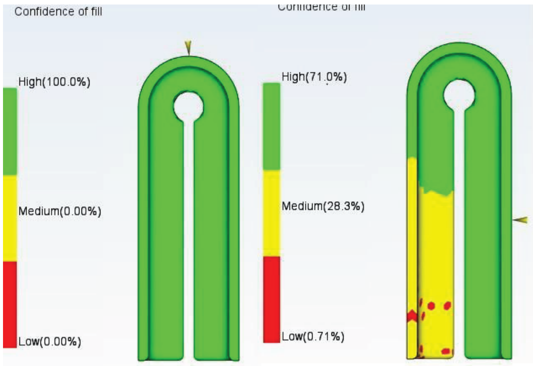

Figure 5. Confidence of Fill Results, GP1 (L), GP2(R)

Further quality prediction tests indicate 84.6% high quality for GP1 while it is coming 50.4% high for GP2. This analysis is done for analyzing the quality of molded part [5]. This analysis is shown in Fig. 6.

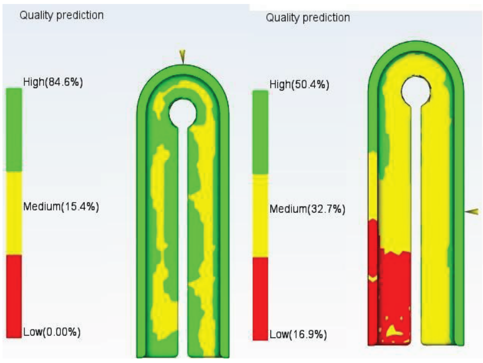

Figure 6. Quality Prediction Results, GP1 (L), GP2(R)

In the similar manner the analysis for fill time indicates that it is taking 4 seconds for filling of the mold cavity from GP1 while it is taking 12 seconds to fill from GP2 under the same amount of pressure. Fig. 7 is representing the filling of the cavity at 3 seconds. The reason behind quick filling from GP1 can be splitting of stream in camber region and filling of both arms simultaneously while in case of GP2 flow stream diverts to fill one arm first and back pressure causes proper filling of second arm. In general practice longer fill time affects overall cycle time of the part molding. Usually keeping small cycle time is desirable for any manufacturing process.

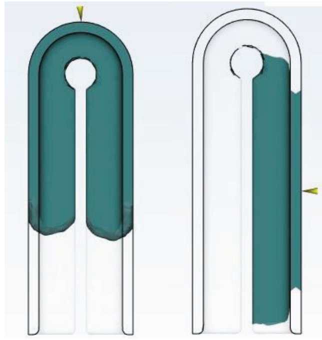

Figure 7. Fill Results in 3 Seconds, GP1 (L), GP2 (R)

Further this work analyzes the cooling quality of the parts. This analysis predicts the way heat will be liberated from the hot part naturally and flow towards outer boundaries of the mold [6]. Fig. 8 represents cooling quality results for both the filling situations. We can clearly observe that the cooling quality is 94.8% high for GP2 filling while it is only 54.3% high for GP1 filling. This is an advantage for GP2 filling which is quite in agreement with previous results. As the cavity is filling with back pressure and taking much time to fill the mold cavity, it is getting cooler at a higher rate than in GP1 filling.

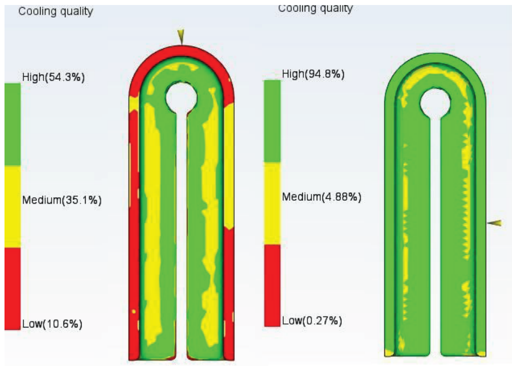

Figure 8. Cooling Quality Results, GP1 (L), GP2 (R)

From all these results it was observed that though there are some advantages with GP2, GP1 is more beneficial in order to obtain good quality components. Cooling quality can be improved in GP1 filling by providing suitable cooling channels. Thus, the work proceeded with GP1 filling.

IV. Die Trial

Based on the consideration in the product design section and outcomes of the simulation, a sample die was machined. Gate position 1 was provided. From the opposite direction of the gate position, vents were provided to ensure proper filling of the mold and no air entrapment into the mold cavity. The machined cope half and drag half of a model mold is shown in Fig. 9. The same is representing all the elements of flow path including air vents. This mold was aligned at a semi automatic injection molding machine, specifications of which are mentioned in Table II. Trial was taken after opening one half of the mold, part obtained after 1st trial is shown in Fig. 10.the output was as per the simulation results.

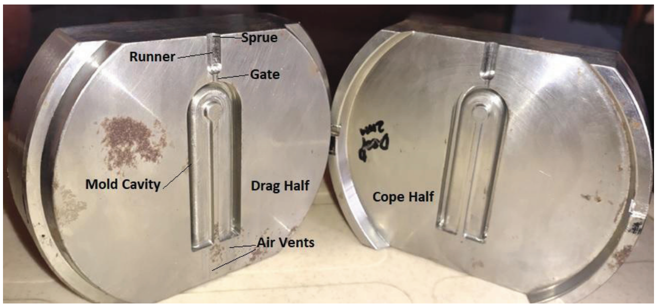

Figure 9. Cope and Drag Half of Model Mold

Figure 10. First Trail (Drag Half Removed)

Parts after trial were observed for camber due to shrinkage behavior. It was expected and the same was discussed in the product design section. Fig 11 indicates two parts adjacent to each other. The left side part is immediately taken out of mold thus it is undergoing solid shrinkage. Here the arms are straight as per mold cavity. But the right-side part is cooled to room temperature and hence it has undergone camber due to shrinkage [7].

Figure 11. Camber, After Cooling to Room Temperature

The main reason behind camber is the solidification pattern in the last to solidify region. Metal accumulation at outer regions easily liberates temperature to become solid while the material remains in liquid form at core. Due to this change of phase in a cross section, camber occurs as an outcome of shrinkage. Fig. 12 is representing final parts after removal from flow path elements and its application.

Figure 12. Manufactured Parts (L) and Application (R)

V. Conclusions

This research work was dedicated to analyzing an existing problem and obtaining a creative solution for that. A designer must analyze the manufacturability of his design. At the same time, he cannot ignore the cost efficiency of his provided solution. For the plastic part considered in this work, simplicity of the design and cost effectiveness are the most shouted benefits. The considerations and the standards followed in this work can be utilized by upcoming researchers for their work in the field of tooling. This work has not done cost analysis and market research to analyze success or failure of the product as it can be part of further extended research. But the work has considered and provided enough justification for representing this part as one feasible solution which is durable and manufacturable at less cost.

Based on the outcome of this research work a proper injection molding die can be manufactured with multiple cavities, cooling provision and mechanism of ejection. This work doesn't claim that the provided solution is the only solution for the problems occurring in the existing clothe pegs. Based on further analysis and expertise in the field of product development, more feasible solutions are also possible. As per the feedback provided after using the part produced, a few possible improvements can be, 1. providing a bigger fillet at the end of the arm and 2. providing depression at the outer surface of the arms for a better grip.

References

[1] Kim, Y.-H., Park, S.-W. and Sawng, Y.-W. (2016), "Improving new product development (NPD) process by analyzing failure cases", Asia Pacific Journal of Innovation and Entrepreneurship, Vol. 10 No. 1, pp. 134-150. https://doi.org/10.1108/APJIE-12-2016-002

[2] J. Bouquet, D. Van Camp, O. Malek, P. Ten Haaf, K. Vanmeensel, B. Lauwers, Aiming for Improved Lifetime of Die and Mold Components through an Integrated Laser Hardening Operation, Combining Machining and a Selective

Heat Treatment in One Setup, Procedia CIRP, Vol. 46, 2016, pp. 541-544, ISSN 2212-8271, https://doi.org/10.1016/j.procir.2016.04.018.

[3] Rupendra, Ch & Vivek, D & Sai, Yuva & Raju, Sudhakar & Jogarao, Bikkavolu. (2020). Design, Analysis and Fabrication of Safe Holder and Cam Dies in Injection Molding. International Journal of Recent Technology and Engineering (IJRTE). 8. 2277-3878. 10.35940/ijrte.F9707.038620.

[4] Youmin Wang,and Guoqing Wang, (2022), “Research on the Injection Mold Design and Molding Process Parameter Optimization of a Car Door Inner Panel”, Advanced Composite Materials and Their Applicationshttps://doi.org/10.1155/2022/7280643

[5] Hongbo Fu, Hong Xu, Ying Liu, Zhaogang Yang, S. Kormakov, Daming Wu and Jingyao Sun, “Overview of Injection Molding Technology for Processing P olymers and Their Composites ”, ES Mater. Manuf., 2020, Vol. 8, pp. 3–23

[6] Neeraj Kumar Jha and Bhavya Sri Tadiparthi, (2019) “Design and Analysis of the Sleeve Ejection System in Injection Molding Die for Trolley Wheel”, CVR Journal of Science and Technology,Vol 17 No 1.

[7] Wu, Y., Gong, Y., Cha, K.J. et al. Effect of microstructures on the shrinkage of injection molding product. J Mech Sci Technol 33, 1357–1363 (2019). https://doi.org/10.1007/s12206-019-0236-y