E-ISSN 2581 – 7957
P-ISSN 2277 – 3916
CVR Journal of Science and Technology, Volume 22, June 2022
DOI: 10.32377/cvrjst2213

Design and Analysis of Injection Mold for Plastic Rivet with Buttress Thread Profile: DFM Approach

Neeraj Kumar Jha[1], Vidyanand Kumar[2]
[1]Assoc. Professor, CVR College of Engineering/ Mechanical Engg. Department, Hyderabad, India
Email : neerajjha.me@gmail.com
[2]Asst. Professor, CVR College of Engineering/ Mechanical Engg. Department, Hyderabad, India
Email: vidyanandk75@gmail.com

Abstract: **Manufacturing is the process of transforming raw materials to useful product. Due to this transformation the finished product used to be costlier than the raw materials. Generally, the entire process is planned to maximize profit. Different attributes help to achieve this goal. The fulfilment of customer needs or use of the product is unalterable, but many other aspects can be adjusted in the focus of ultimate profit. Selection of material, process selection and design of the part are few among such attributes. Injection molding is one of the widely used processes to process polymers, especially thermoplastics to any required shape. This paper aims to make rivets with Polypropylene (PP) using the injection molding process. The paper proposes a new design for plastic rivets with modified buttress profile. Simultaneously this work checks the suitability of making such rivets by injection molding process. It was found after the software simulation that such new product can be easily molded without any considerable defects. The software implemented in this works are, UG Nx 12.0 for solid modeling of the parts, Autodesk Moldflow Adviser 2019 for flow analysis and UG Nx 12.0 Design simulation for stress analysis.**

Index Terms: **Injection molding, Buttress Threads, Rivet, Polypropylene, Plastic Mold, Tool, Inserts.**

I. Introduction

Design for manufacturing (DFM) is a tool that is used during product development phase with the aim of ease of design and manufacturing of the component [1]. It is prudent to use DFM principles while designing a new product for the assurity that not just the need of the product is fulfilled but the design and manufacturing processes of the product is also done in the most efficient way [2]. Implementation of DFM principles ensures, standardization, quality improvement of product, reduction of number of parts and steps in manufacturing as well as manufacturing cost reduction. Similar and improved manufacturing philosophies that evolved from DFM are Design for Assembly (DFA), Design for Manufacturing and Assembly (DFMA), Design for Excellence (DFX) etc. All these terms collectively aim at design and manufacturing excellence without compromising with the profit. In general, there are no set rules for implementation of DFM. It depends on the manufacturing process or the set of operations to be followed for a product, in order to fulfill its aim of decreasing number of processes and steps to gain maximum profit. As minimum number of parts and optimized assembly sequence are considered as healthy DFA practices, in the same way the best process selection, the best material selection, adequate tolerance selection and the least process sequence can be considered as pillars of DFM. A suitable process selection is an unavoidable step in the production process. Many activities in process planning are influenced by the selection of manufacturing process selected for the manufacturing of a part [3]. Though a part can be prepared by various manufacturing processes, a good designer takes decision based on parameters like estimated time, cost involved, skill involved, expected quality etc.. Correct selection of material is very important for operational aspect of a product. Additionally, it influences the manufacturing operation selection also. Different materials are treated differently, utilizing adequate manufacturing processes. Time and energy consumed in the processing, affects the cost of production. Adequate tolerance selection is another important factor in DFM implementation. Close tolerance dimensions keep the cost of production on the higher side, which is not needed unless it is required. Thus, open tolerances should be given as maximum as possible for the parts to control the cost of manufacturing. Contribution of least process sequence can never be avoided in DFM as well as in product process planning. It is evident that more processes involve more inventory and resources, thus making the entire processes costlier. Many revolutionary changes are made worldwide in the consumer product industries. The outcomes were phenomenal and resulted in cost saving, time saving, sustainable product development etc. Automobile industries are the most benefited sector from the implementation of DFM principles. Advancement in material technology and manufacturing processes has added extra dimensions in the implementation of DFM principles. Evolution of different grades of plastic and their processing methods have made exceptional changes in the field of design and manufacturing. A plastic material, due to its unique properties is becoming the most suitable material for many engineering applications. One of the best properties of plastic is its moldability, which makes it more suitable to come to a usable shape [4]. Usually, reusable grade of plastic i.e., thermoplastics are processed by injection molding process. Nearly 70% of total plastic products are made by this process [5]. The process utilizes a metal mold with shaped cavity in it. Hot plastic in semi solid phase is injected to the shaped cavity. The mold is opened, and the part is ejected from the cavity after cooling and solidification of the shot volume. The process is simple to adapt and opted for mass production of small to big size plastic parts with simple to complex profiles. The process is simple and can be automated also but, if the process parameters are not controlled properly the defects will be

Received on 08-04-2022, Revised on 18-04-2022, Accepted on 25-05-2022.

E-ISSN 2581 – 7957
P-ISSN 2277 – 3916

CVR Journal of Science and Technology, Volume 22, June 2022
DOI: 10.32377/cvrjst2213

unavoidable. Recent advances in analysis and simulation techniques are useful for designers to take care of defects [6]. Utilizing these techniques one can easily estimate defects without wasting time and money. Autodesk Moldflow Advisor is one such simulation software which replicates the actual injection molding process. It utilizes actual process parameters viz. injection pressure, mold and injection temperature, material properties etc. to give output like cycle time, cooling time, defects visualization and molding quality corresponding to a part model [7, 15].

This paper attempts to redesign plastic rivets and see its feasibility to be manufactured by the injection molding process. Rivets are mechanical members which are used to create semi-permanent joint between sheets or plates. Metallic rivets were much in use till there were limited welding processes available for joining thick metal plates. Development of welding technology limited the use of rivets but the place of rivets in engineering applications is unavoidable. After revolution of plastic as engineering materials, plastic sheets were being made by the calendaring process and rivets were used for joining them. In various consumer products, especially in automobiles plastic rivets can be seen joining plastic parts together. Advantage of using rivets is that they hold the joining parts permanently but if there is need of detaching the parts, rivets are broken, and the joined parts remains unharmed. Parts of metallic rivets are formed by joining. Plastic rivets implement snap fits for their joining. In later portion of this paper product design of plastic rivet is explained, further with help of a sample mold its simulation for injection molding process is done. Its operation and stress analysis are also performed. There is crucial need of such innovative approaches in order to simplify the production process.

II. Product Design

A. Product Modifications

The existing plastic rivets are made in two parts which interlock with each other after they are inserted through the hole made in the parts to be joined. Snap fits are widely used for interlocking across the various existing models. In this research work, buttress profile was used for the interlocking instead of snap fits. The purpose is of utilizing high load carrying capacity of the profile to prevent opening or the rivet. The terms buttress profile here must not be wrongly considered as buttress threads. Only the cross-section is being used here but not the helical profile. Buttress profiles are asymmetric profiles used in buttress threads, preferably used when there is need of bearing high load in one direction only. Fig. 1 shows a standard buttress profile which is as per 45° DIN 2781 standards, while Fig. 2 and Fig. 3 (All dimensions are in mm) indicates the part drawing with modified buttress profile which is used for this work. The first part with internal buttress profile is termed as body while its counterpart with external buttress profile is termed as core. The overall dimension of the rivet is taken from standard dimension of a snap head rivet with body diameter 10mm. The snap head profile is in the same standard proportion.

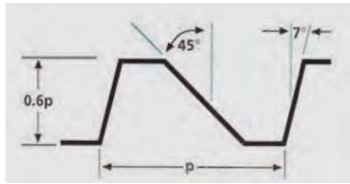

Figure 1. Buttress profile (p: pitch)

The pitch value (p) is considered 1.5 from standards. The internal diameter of rivet body and the external diameter of rivet core are taken as empirically compatible dimension.

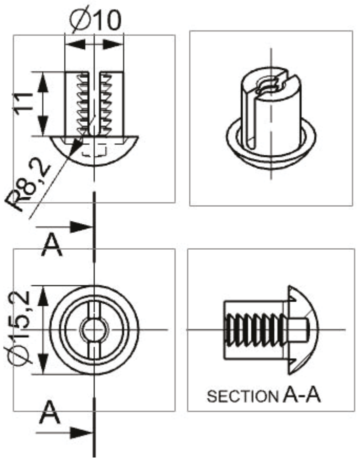

Figure 2. Snap head rivet body drafting

Apart from the modified buttress profile, few other design features can also be visualized. The body is profiled by two slotted recesses on it. This allows proper insertion of core in the body while applying them against a joint

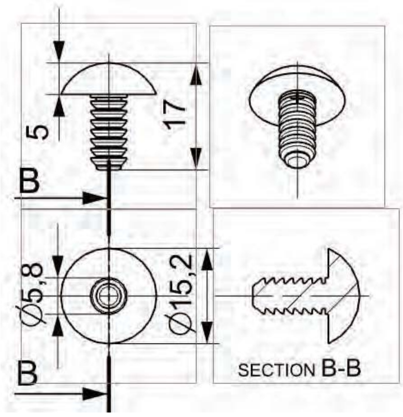

Figure 3. Snap head rivet insert drafting

B. Embodiment Design

For any product the process parameters lead a crucial part in determining its feature details and ultimate form. Thus, a product's final form can be decided after many critical decisions are taken. The embodiment design is primary and important phase of product design. At this phase important decisions like material selection, process selection, component size etc. are taken. The part is modified accordingly and further process planning for corresponding to a part is done [8]. The material for the part was decided as Polypropylene (PP) for instance, due to its usually low range of processing temperature and adequately high range of strength. Other suitable materials can be HDPE as well as PVC for this application. Other advantages of using these materials can be reusability, good surface finish and relatively low density. It is evident that these grades of plastics can be conveniently processed by injection molding process. Other important properties for the point of view of process parameters are shown in Table 1.

TABLE I.
MATERIAL SPECIFICATION , POLYPROPYLENE (PP)

Density	946 kg/m³
Melting Point	160 °C
Formula	$(C_3H_6)_n$
Type	Thermoplastic
Flexural Strength	40 N/mm²
Shrinkage	1-2.5% mm/mm
Tensile Strength	32 N/mm²
Injection Temperature	32-36 °C
Heat Deflection Temperature	100 °C
Specific Gravity	0.91

In the phase of embodiment design the designers conventionally choose material and processes based on their experience or it is done based on the experience of the manufacturing engineer. Once it was decided that the part will be made from PP using injection molding process, the part should be made suitable for injection molding process. Thus, as per DFM principles adequate fillets, draft etc. should be given. The plane of symmetry is considered as parting line. On all the faces, which are in the direction of ejection the adequate drafts are to be given. As the part is mostly curved body thus, no drafts are needed. Fillet is an important concern for proper filling of the mold. On all necessary surfaces adequate fillets were given. This can be observed from the drawings as well as from the solid model shown in Fig. 4. In the figure the left side image shows core and body engaged in fully lock position. In the right-side image, body is kept 50% translucent for better visibility. Empty space under snap head was an attempt to reduce material accumulation so that shrinkage cavities can be avoided. This gap additionally acts as compression member for tight fitting of rivet. This can be ignored in the manufacturing phase.

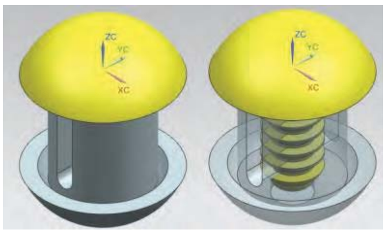

Figure 4. Solid model: snap head rivet

The thick mass analysis is shown in Fig. 5. Minimum wall thickness is considered as 5mm. Green color is indicating more than 5mm thick wall where less than 5mm thick wall is indicated by red color. Thick mass regions may attract shrinkage cavities which can be taken care by design as well as process parameters. Its solution as per design is making weight reduction pocket, as it is made under snap head of the body. Shrink cavities can be taken care by high injection pressure as well as by packing pressure. For specific component profiles, proper cooling can also help to overcome shrink cavities.

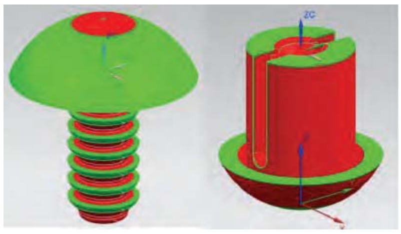

Figure 5. Thick mass analysis

The proposed design is capable to join variable thickness of sheets. It is observable that gap between head of body and the insert is 10mm but, it can accommodate sheets with total thickness 1.5mm to 15mm. The gap between each step is 1.5mm. The gap between heads can be made 15mm when few steps are effectively in contact. This functionality is unavailable in most of the existing plastic rivets. To accommodate length, extra part can be sliced. Snap head also can be rubbed off if there is need of flat surface. By considering all the aspects of design as per ease of manufacturing the part without defect, there are few changes made in the body design. It is indicated in Fig. 6. The gap under snap head is filled and a through hole is provided to prevent mass accumulation. The same will help in positioning the insert to achieve internal buttress profile.

Figure 6. Final design changes in rivet body

III. VALIDATION

The designed component must be validated to check if it is meeting the design and manufacturing expectation. Stress calculation supported by design simulation UG Nx 12.0 results was done for validation. Similarly, its manufacturability was predicted by Mold flow adviser.

A. *Stress Analysis*

A rivet fails mainly in three situations, viz., shearing, tensile failure and crushing. While installing metallic rivets against metallic plates, rivets get work hardened and their strength and hardness increases. This hardened rivet when pulled along rivet plate, it causes elongation of hole in the plate in which rivets are installed. Such failure of riveted joints is termed as crushing or bearing failure. Unlike metals, plastic shows least or negligible chances of work hardening. Thus, crushing failure of plastic rivets is not being given much importance in this work. Shearing of the plastic rivets is also not so frequent unlike metal rivets. Reason behind this phenomenon can be existence of plastic rivet in two pieces. Metal rivets used to be in one part. Thus, along a plane they may get sheared much easily than compared to plastic rivets. By ignoring chance of failure of plastic rivets by shearing failure also this work is focusing only on tensile failure of the plastic rivet which is much in agreement with the design modification of the proposed part. Further two aspects were checked. One is checking of the maximum load this rivet can take, and at that load whether the teeth are deflecting to get detached. The maximum stress occurs at least cross-sectional area. There are two places where smaller cross-sectional area is possible. One is body and another is smallest diameter of core as shown in Fig. 7.

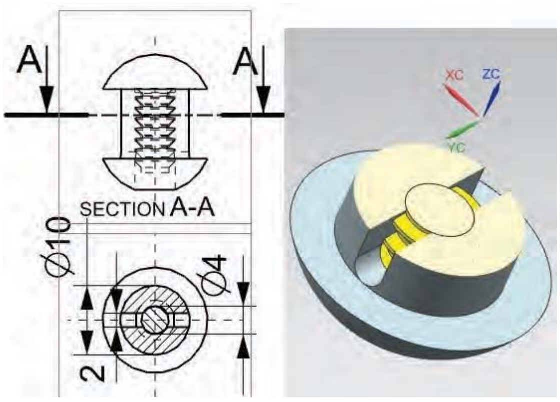

Figure 7. Cross sectional areas to be considered for tensile strength

By visual inspection smaller area can be observed but the same is calculated in (1) and (2). From these calculations it is predictable that, more stress is going to develop on core. Ultimate load that can be applied on this area also is calculated in (3). This is the maximum amount of load which the designed rivet can take before breaking. Displacement analysis of buttress profile is done on UG Nx 12.0 design simulation software. The same is shown in Fig. 8. Ten node tetrahedral elements were created with 2mm size elements for meshing. Left portion of figure indicates the constrained face and the faces on which load is being applied. It is evident that the load will come from the similar faces of rivet body. Right side image in Fig. 8 represents the deformation result. The load applied for the analysis is 200N. This load range is empirically taken, assuming half of the ultimate load. It is observable that small deformation has occurred, but the profile is maintained. Thus it can be assumed that due to deformation the teeth will not disengage.

$$\text{Cross section area of Body} = [\pi(10^2-4^2)/4] - 2(3\times 2) = 53.97\text{mm}^2 \quad (1)$$

$$\text{Cross section area of Core} = \pi(4^2)/4 = 12.56\text{mm}^2 \quad (2)$$

Maximum tensile force can be applied on core
= (Tensile strength of PP) X (CS area of core)
= 32 X 12.56 = 402 N (3)

This work is focusing only on the tensile failure. Failure of rivets by combined effect of stresses as well as analyzing the efficiency of plastic rivets is beyond the scope of this work.

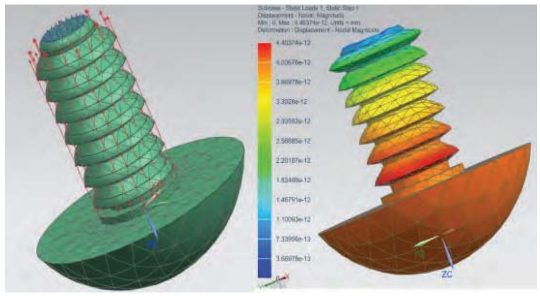

Figure 8. Constrained face and faces with applied load along with the displacement results

The displacement result is quite in agreement with the design expectations. Further the manufacturability of the part should be analyzed.

B. Process Simulation

To check the compatibility of the part to be made by injection molding processes, this work utilized Autodesk Moldflow Adviser 2019 software. This is widely used software in industries for the purpose of injection molding processes simulation. It is efficient and cost-effective tool for process simulation during component development. Thus, it helps in reducing product development time and cost. Before running the simulation on the designed model, few input parameters related to the process are provided. For this purpose, this research work has taken parameters from an injection molding machine. For reference these parameters are indicated in Table II.

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

Shot Capacity	2 – 45 gms / shot
Plunger Diameter	25 mm
Stroke Length	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

The process simulation helps in predicting defects and other process related issues. This work was aimed at checking if the proposed design is manufacturable or not. From the standard calculations the gate and runner diameters were calculated [9]. In single impression, core as well as body of the rivet were planned. Edge feeding was finalized along plane of symmetry. The expected shot volume for the part cavity including flow path elements is estimated from the software as 15.3gm, which is well within the shot capacity of the machine. After entering the needed input parameters, the flow analysis simulation was conducted and various useful outputs were obtained. Confidence of fill is indication of how conveniently the planned cavity can get filled under given process parameters. There is high probability of filling which is shown in Fig. 9. It is observable that sudden sections changes, as per modified buttress profiles are also not an obstacle in proper filling of the mold. This can be considered as outcome of DFM related modification and the design changes made accordingly [14].

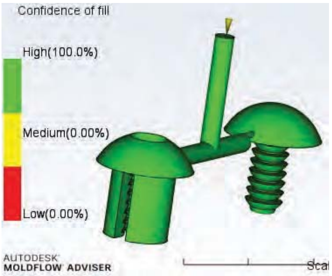

Figure 9. Confidence of fill result

Similarly, another important prediction is shown in Fig. 10, which displays expected time which the part will take to solidify. Time to reach ejection temperature is sum of fill time and solidification time. It helps in estimation of cycle time. In the figure, mostly the part is blue i.e. in 8 seconds majority of the part is solidifying. Some thick mass regions of core are taking time in solidification. Usually, 60% of the cycle time is cooling time. A designer always aims at decreasing this time to as least as possible [10, 16]. It is significant to mention that cooling channels were not planned here.

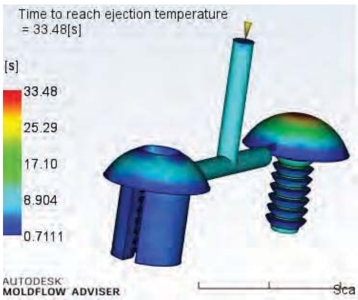

Figure 10. Time to reach ejection temperature result

As stress will be involved in rivet while in application, it is important to check weld lines. These can be considered as structural deficiency which comes into existence due to merging of flow fronts [11]. In Fig. 11 the weld line results are displayed. Due to empty space provided in the rivet body, slight weld line of small angle is appearing in snap head. It is not a problem in this case due to its small quantity in low stress region.

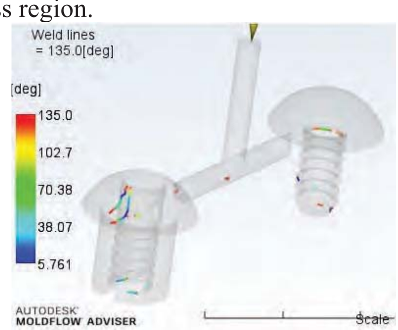

Figure 11. Weld line result

As expected from embodiment design of part, there is minute sink mark in thick mass region of the part, as indicated in Fig. 12 as sink mark results. Maximum possible mass reduction in terms of design were done. Thus, this sink mark can be possibly handled in actual process by applying packing pressure [12].

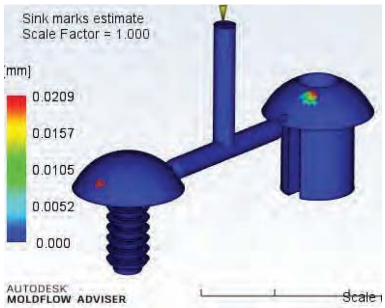

Figure 12. Sink mark result

Another important result is shown in Fig. 13. This is cooling quality result. It indicates the region where heat will remain for the longer time during solidification. Portions of the parts, which are appearing in green will solidify earlier and the portions indicated in red will be the last to solidify. This result further helps in planning of cooling channels for maintaining temperature of the metallic mold used in the injection molding process [13].

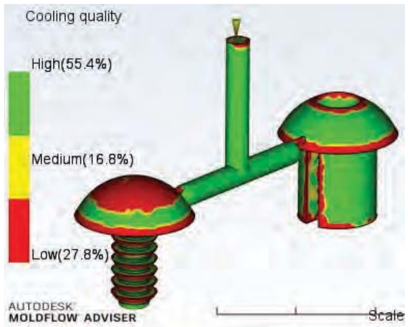

Figure 13. Cooling quality prediction

IV. The Mold

Based on the simulation results, a mold was modelled for making the samples of the designed rivets with buttress profile and the same is shown in Fig. 14. The mold is planned according to the machine specification of which is considered for the flow analysis. The machine is with vertical injection and horizontal clamping facility. In the mold, cavity was planned accordingly. Dimensions of the mold were taken as per die design parameter handbook [9]. In the figure, one half of the mold is made translucent so that flow channels, mold cavity and placement of insert can be seen. For better understanding rivet core and body is placed in their respective mold cavities and rivet body is made further translucent to see the insert placed in the cavity. This insert is to be placed in the mold before injection so that internal buttress profile of the rivet body can be obtained. Machining of the mold cavity can be done by end milling. The insert is to be obtained by turning for ease of manufacturing. This is plan of a model mold. Thus, cooling channels are not installed and the same is not considered in the analysis. After filling the plastic material will solidify around insert. Thus, after ejection the insert will be removed manually from the rivet body. The same insert will be placed in the mold before next filling. In actual die which will be made for the production purpose, ejection of insert can be mechanized. As the mold is split in two halves, there is provision of four guide bush and pillars to prevent mismatch of both the halves while closing and filling.

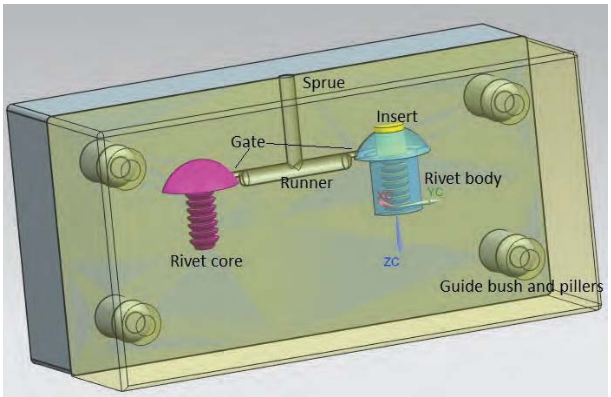

Figure 14. The designed mold

V. Conclusions

In this research work an attempt was made to modify the design of existing plastic rivets and check its manufacturability. Adequate load analysis and mold flow analysis was performed which resulted in assurity that the design modifications are not only making the product capable to sustain loads, but the design changes are making the product manufacturable also. This work gives clear methodologies for a new product development to the young researchers. Designing a mold for this component for industrial purpose can be further part of research. Planning optimum flow path for a multi cavity mold can also be taken as future scope of work. Current innovations in material and manufacturing technologies are making the manufacturing of designed parts simpler and more cost effective. Thus, comparing the manufacturing of a part by different manufacturing process also can be added in further analysis. For example, the parts taken in this work can also be manufactured by 3D printing technology. Usually design of a part is finalized by considering a manufacturing process, but a successful design not just contributes to manufacturing but to overall profit of the entire production process. There are no set rules for implementation of DFM principles. It all depends on the designer to take profitable decisions in process planning.

References

[1] Boothroyd, G., Dewhurst, P., Knight, W. (1994). Product Design for Manufacture and Assembly. New York: Marcel Dekker Inc.

[2] Barbosa, G.F, J. Carvalho, Design for Manufacturing and Assembly methodology applied to aircrafts design and manufacturing, IFAC Proceedings Volumes, V- 46, Issue 7, 2013, pp.116-121, https://doi.org/10.3182/20130522-3-BR-4036.00044.

[3] Darshak Desai, Bhavikkumar Nileshbhai Prajapati, "Competitive advantage through six sigma at plastic injection molded parts manufacturing unit: a case study", International Journal of Lean Six Sigma, https://doi.org/10.1108/IJLSS-06-2016-0022

[4] Farotti, E.; Natalini, M. (2018). Injection molding. Influence of process parameters on mechanical properties of polypropylene polymer. A first study.. Procedia Structural Integrity, 8, 256–264. doi:10.1016/j.prostr.2017.12.027

[5] Naol Dessalegn Dejene, Sololo Kebede Nemomsa, "Design for Manufacturing and Assembly- Review," World Academics Journal of Engineering Sciences, Vol.7, Issue.3, pp.60-67, 2020.

[6] Ming-Tsan Chuang , Yung-Kuang Yang & Yun-Hsiang Hsiao (2009) Modeling and Optimization of Injection Molding Process Parameters for Thin-Shell Plastic Parts, Polymer-Plastics Technology and Engineering, 48:7, 745-753, DOI: 10.1080/03602550902824630

[7] Sateesh, N.; Devakar Reddy, S.; Praveen Kumar, G.; Subbiah, Ram (2019). Optimization of injection moulding process in manufacturing the flip cover of water meter. Materials Today: Proceedings,S2214785319337721,doi:10.1016/j.matpr.2019.11.089.

[8] Proceedings of DETC'04 ASME 2004 Design Engineering Technical Conferences and Computers and Information in Engineering Conference September 28-October 2, 2004, Salt Lake City, Utah, USA.

[9] Technical committee, *Technical Directory on Design and Tooling for Plastics*, Central Institute of Plastic Engineering & Technology. pp. 43-64, 1970.

[10] SÃ¡nchez, R.; Martinez, A.; Mercado, D.; Carbonel, A.; Aisa, J. (2021). Rapid heating injection moulding: An experimental surface temperature study. Polymer Testing, 93, 106928–. doi:10.1016/j.polymertesting.2020.106928

[11] Mehdi Moayyedian, Kazem Abhary, Romeo Marian, Improved Gate System for Scrap Reduction in Injection Molding Processes, Procedia Manufacturing, Volume 2, 2015, Pages 246-250, ISSN 2351-9789, https://doi.org/10.1016/j.promfg.2015.07.043.

[12] Neeraj Kumar Jha, G. Bharath Reddy, Vidyanand Kumar, "*Design and Process Analysis of Single Cavity Injection Molding Die for Plastic Wing Nut*", CVRJST, Vol. - 20, 2021.

[13] Rashid, O.; Low, K.W.Q.; Pittman, J.F.T. (2020). Mold cooling in thermoplastics injection molding: Effectiveness and energy efficiency. Journal of Cleaner Production, 264(), 121375–. doi:10.1016/j.jclepro.2020.121375

[14] Neeraj Jha, G. Avinash, V. Siddharth, "Design and structural analysis of plastic chain link with polypropylene and polyoxymethylene material", November 2020, Materials Today: Proceedings 38(1)

[15] Huang, Yu-Chen; Tu, Jui-Che; Kuo, Kuo-Pin (2017). Establishing sustainable design and development for plastic mold under product service system. Advances in Mechanical Engineering, 9(7), 168781401770969–. doi:10.1177/1687814017709699

[16] Singh, Gurjeet; Verma, Ajay (2017). A Brief Review on injection moulding manufacturing process. Materials Today: Proceedings, 4(2), 1423–1433. doi:10.1016/j.matpr.2017.01.164