E-ISSN 2581 – 7957
P-ISSN 2277 – 3916
CVR Journal of Science and Technology, Volume 26, June 2024
DOI: 10.32377/cvrjst2619

Design and Analysis of Plastic Injection Mold for Hexagon Socket Head Cap Screw

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

Abstract: **Plastic is a majorly used material for making customer goods. It is appealing to customers as well as manufacturers. Properties like durability, light weight and moldability make it a suitable choice for consumer goods which are made by mass production. Plastic materials are soft, but they possess adequate strength to be utilized for different engineering applications. After the development of various grades of plastic, it has replaced metals from different engineering applications in the recent past. This article aims at analyzing the suitability of making Hexagon Socket Head Cap Screws (HSHCS) out of a plastic material by injection molding processes. These screws are widely used as threaded fasteners at different places. They are preferably made of ferrous metals and possess semi complex design. They are mass produced by following a series of manufacturing operations. This article will observe the manufacturability of HSHCS by injection molding process. Based on the observation a model mold will be prepared to make HSHCS. The software used for modeling the parts is Autodesk Fusion 360 and that for the simulation of process parameters is Autodesk Moldflow Adviser 2023.**

Index Terms: **Hexagon Socket Head Cap Screws, plastic mold, manufacturability, threaded fasteners, injection molding, simulation.**

I. Introduction

Rapid growth in materials and manufacturing technology is making all engineering challenges achievable. Design innovations are contributing to ease the manufacturing. New materials are allowing engineers to think beyond conventional. After the invention of polymers, they are being conveniently used to meet specific technical needs. Various plastic processing techniques are developed for processing tailored polymers to different shapes [1]. Injection molding is one such process. This process utilizes facilities to convert plastic material to semisolid phase and pressurizes it to a shaped cavity. The cavity possesses the shape of the required product. It is made of a metallic mold, often called plastic mold, or die. This mold is designed by following a standard set of procedures. Getting a defect free product from this process depends on controlled process parameters and effective design of die. The design of this mold can be simple, or complex based on complexity possessed by the component. A component with more undercuts and pockets may need inserts in the die. Sometimes for complex part multidaylight molds are needed. The decision about number of cavities, gate location, runner diameter, ejector pin location etc. is also part of the design process. Controlling the process parameters means controlling injection temperature, cooling time, injection pressure and cycle time. The injection molding process is the most suitable for mass production of plastic parts irrespective of size and shape of the part. Current trend in manufacturing justifies 3D printing for making various plastic parts. But it is evident that the parts made by 3D printing have large lead time [1]. Thus, the parts which are very complex and nearly impossible to make by injection molding are only made by 3D printing. Threaded parts are usually considered under the category of complex parts. Usually, threaded parts are obtained by machining process. But it is unique advantage of injection molding process that a part obtained by injection molding can have threaded profile on it [1, 2].

HSHCS are threaded parts. They are a type of screw, which are designed and made with internal hexagonal drive formed into the head while the outer edge of screw head remains round. They are actuated with a hex key often called Allen key for tightening or loosening purposes. They differ from conventional screws and bolts in many ways. They do not need a nut to be positioned for locking. They utilized internal thread of to be joined parts like a screw does. HSHCS possess many advantages over their conventional counterparts. The hexagonal socket enables the screw to resist cam-out better than other similar drives. This is also evident that HSHCS of the same size enable us to gain higher torque and clamping force. When applied, the whole head of HSHCS sinks into the workpiece, thus the surface of workpiece remains projection free and appears good aesthetically. They are not very easy to disassemble, as they need a special type of tooling for actuation. They have a wide range of applications, especially in tooling and manufacturing industries. Usually, to join multiple plates and parts together HSHCS is used. They can be observed beneath the surface of external plates of various tools viz. forming tools, jigs and fixtures, plastic molds, die casting dies and machine tools. They are made of metal, preferably alloy steel following a sequence of manufacturing processes. A rolled bar is first cut to a calculated length which goes through heading operation to form head of the HSHCS. At the same time the hexagonal recess in the head is also formed. Further threading is done on its body either by thread cutting operation or by the thread rolling process. Additionally, the side face of the head is knurled for better grip during casual actuations. There is a possibility of another combination of operation also to manufacture SHCS but the one discussed has the smallest lead time.

This article made an attempt to make HSHCS out of a suitable plastic material by injection molding process. The purpose of developing HSHCS out of plastic was to gain benefit of single station manufacturing there by reduced manufacturing cost and implementing them for joining metal parts in low load conditions. Further this article

Received on 27-01-2024, Revised on 26-02-2024, Accepted on 27-03-2024.

CVR Journal of Science and Technology, Volume 26, June 2024
DOI: 10.32377/cvrjst2619

explains the product design phase in Section II. Section III deals with design calculations. The discussion about process parameters and simulation of the process is in Section IV. Section V is about the trial of the manufactured die. Finally in Section VI the conclusion of this work is demarcated.

II. Product Design

Product design is the most important element of mold design [2]. An effective product design ensures simple mold design and overall profitability of process [3]. Smooth process flow and defect free products are outcomes of product design [3, 4]. The 2D drawing and solid model of M8 X 1.25 HSHCS considered for this work as well as its solid model is shown in Fig. 1. HSHCS dimensions include height and diameter of the head, shaft diameter, length of threaded shaft. The dimensions are influenced by BIS 2269: 2006, ISO4762: 2004.

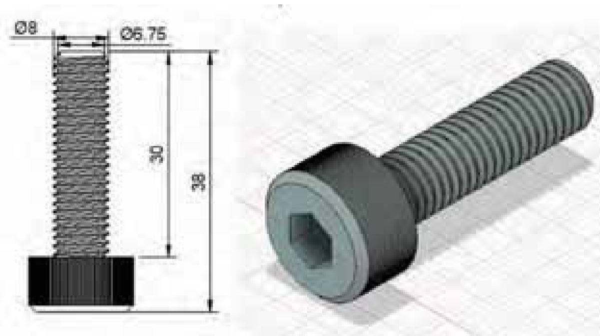

Figure 1. 2D Drawing of HSCHS and Solid Model
(All dimensions are in mm)

The part was modeled by considering metallic part. It has hexagonal recess on head and the side face of the head has straight knurling. The part was keenly observed for its effective shaping by injection molding process. Its draft analysis was done to plan its profiles in the halves of the mold. The draft analysis is shown in Fig. 2. According to vertical axis (Z Axis, mold opening direction), top half profiles are shown in green color, bottom half profiles are shown in blue color and straight faces are shown by red color.

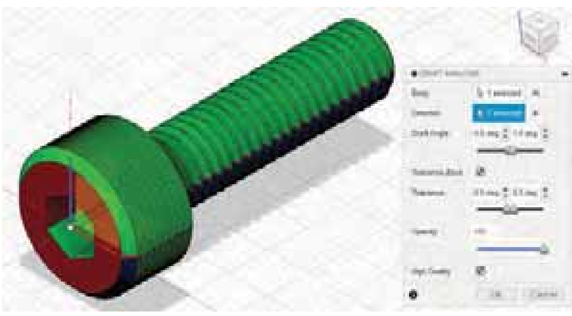

Figure 2. Draft Analysis

Availability of one color in other half indicates possibility of undercut. This indicates trouble in achieving such profiles while opening of mold. It can be observed that there are undercuts in hexagonal recess region and in the knurling. For ease of die manufacturing, it was decided that the hexagonal recess will be obtained with a side insert and the knurling will be avoided from side face of the head. Further observations predicted that entire head can be taken as insert while the threaded shaft will come in the two halves of the mold. The final model of HSHCS with representation of parting planes is shown in Fig. 3. Blue color plane shows splitting of insert to make socket head with body. By this way straight knurling on the side face if the head is possible. But considering the cost of mold making, knurling is avoided from the side face of head. The same can be observed in Fig. 3. The plane in orange color shows splitting of two halves of mold to split threaded shaft.

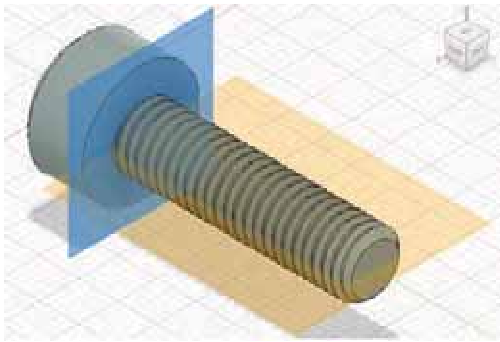

Figure 3. Parting Planes

Material selection can also be considered as part of the product design [5, 6]. This research considered polypropylene (PP) for plastic molding of HSHCS due to its own advantages. A few of the important properties of PP is mentioned in Table I. Finally, 1.5% shrinkage allowance is added to the model as per ISO 294-4 for thermoplastics, considering polypropylene material.

TABLE I.
Material Specification (PP)

Material Properties	PP
Density	946 kg/m³
Softening Point	150 °C
Formula	$(C_3H_6)_n$
Shrinkage	1-2.5%
Tensile Strength	21-37 N/mm²
Injection Temperature	260 °C
Heat Deflection Temperature	120 °C
Mold Temperature	60°C

III. Design Calculations

Before proceeding for simulation, it is necessary to know about some important aspects of a mold viz. number of cavities, shot volume, clamping capacity. These parameters directly influence the process simulation. In Table II common specifications of an injection molding machine is indicated. These parameters will be considered as references for design calculation as well as for the simulation. This is a vertical injection horizontal clamping injection molding machine.

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

Shot Capacity	2 – 45 gms / shot
Plunger Diameter	25 mm
Stroke Length	450 mm
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 – 450 mm

A. Number of cavities

Decision about the number of cavities in a mold is prime step for the mold design. It is directly influenced by the number of parts to be produced. A lower number of cavities refers to longer lead time and at the same time a greater number of cavities can give defective product if the design is not aligned with the process parameters. The number of cavities is also planned as per space available in the machine between tie bars as it affects the size of the mold.

There are different methods to calculate the number of cavities for a mold [7]. Number of cavities based on shot capacity can be obtained as shown in (1).

$$Ns = \frac{0.85\ Ms - Rw}{Cw} \quad (1)$$

Where,

Ns = No. of cavity based on shot capacity
(Based on 85% of rated shot capacity)
Ms = Rated shot capacity of machine (gm/shot)
Rw = Weight of all sprues and runners (gm)
Cw = Component weight per cavity (gm)

$$Ns = \frac{0.85X18 - 2.15}{4.75} = 2.77$$

The rated shot capacity is considered from the usual shot capacity on which the considered machine works. As per need it can be increased further. Other values are taken from the solid model. The number of cavities is coming out to be approximately three. But considering proper filling of the cavities this research has considered two cavities.

B. Shot Volume

Shot volume is volume of material sufficient to fill all the cavities of the die for one shot. Equation (2) represents all the necessary elements considered for the calculation of shot volume [7].

$$Sv = Vs + Vr + Vg + Vc \quad (2)$$

Where,

Sv= Shot volume (mm^3)
Vs = Volume of sprue (mm^3)
Vr = Volume of runner (mm^3)
Vg = Volume of gate (mm^3)
Vc = Volume of mold cavity (mm^3)

$$Sv = 44 + 1285 + 24 + 9500$$
$$=10853\ mm^3$$

All these values were conveniently taken from the modeled part. Considering the two cavities and the runner system accordingly, the shot volume is coming as 10.85 gm (10853 mm^3). The actual needed material is going to be slightly more than this because shrinkage is also to be considered. It is observable that the required shot volume is well within the shot capacity of the machine.

C. Calculation of Clamping Capacity

Adequate clamping force is needed to hold the split mold so that they will not open due to injection pressure. Equation (3) represents all the necessary elements considered for the calculation of shot volume [7].

$$Cf = PPA\ X\ \frac{1}{3}(IP) \quad (3)$$

Where,

Cf = Clamping capacity (kg)
PPA = Plan projected area of mold (cm^2)
IP = Injection pressure(kg/cm^2)

$$Cf = 6.72\ X\ \frac{1}{3}(80)$$
$$= 179.2\ kg$$

The required clamping force is also within the clamping capacity of the machine. Utilizing outcomes of calculations and product design, planned flow path is shown in Fig. 4. The same outcomes are utilized in process simulation.

IV. PROCESS SIMULATION

The process simulation provides the necessary information to prepare the mold as per process parameters for smooth flow of process to get defect free products. It additionally helps in finding proper gate location for material injection, positions of vents, fill time, flow behavior and possibility of defects.

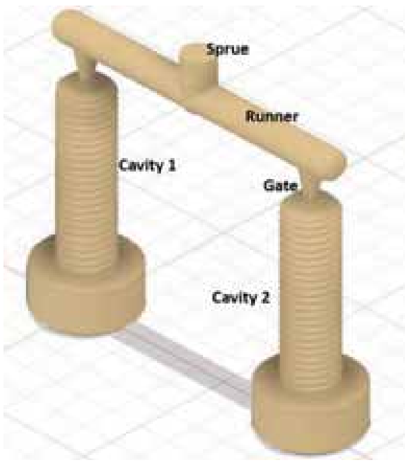

Figure 4. Model of Mold Cavity for Flow Analysis

Considering design standards and values obtained by design calculations from the previous section, a doble cavity mold with flow path was modeled and the same is represented in Fig. 4. The elements of flow paths are labeled. The circular cross section of flow path elements ensures proper flow of melt through the mold [8]. Though

E-ISSN 2581 – 7957
P-ISSN 2277 – 3916

DOI: 10.32377/cvrjst2619

other cross sections are also used for convenience in machining. The gate is usually maintained with least cross section among all the elements of the flow path. It prevents back flow of melt after injection stops. The injection location in HSHCS cavity is planned by considering multiple aspects. Part symmetry plays a major role in this decision. Natural draft available in the part profile, selected parting plane and mold opening directions are other factors affecting selection of the injection location. For mold flow analysis the proper boundary conditions viz. mold temperature (60°C), injection temperature (260°C), injection pressure (4MPa) etc. corresponding to PP were provided as per discussion in previous sections. Mold flow analysis was performed on the model shown in Fig. 4 and the obtained results from Autodesk Mold Flow Adviser 2023 are discussed further.

A. Temperature at Flow Front

In the process of mold filling, flow front temperature should not change beyond 2°C to 5°C [9]. Larger difference represents long injection time. Thus, there might occur solidification of flow front before complete filling of mold. The obtained result is represented in Fig. 5. The difference of maximum and minimum temperature is within the acceptable range.

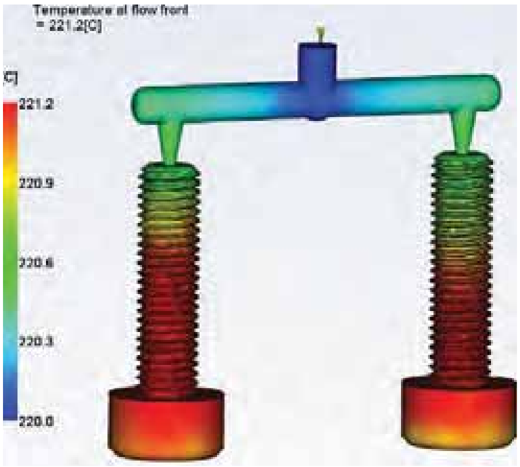

Figure 5. Temperature at Flow Front

B. Weld Lines

The weld lines in injection molding indicate the region where two flow front meet. They are very critical to be examined as they are potential regions of defects [9, 10]. Especially for the parts like HSHCS, weld lines must be avoided for the durability viewpoint. The obtained weld line result is represented in Fig. 6. There were negligible weld lines observed. Longer weld lines around any cross section can be the spots of crack initiation and propagation. Correct gate location can be a reason behind positive weld line result. They can also be avoided by maintaining adequate injection pressure and temperature at the time of mold filling.

Figure 6. Weld Lines

C. Air Traps

An air trap occurs when the melt accumulates and compresses a bubble of air or gas in the flow fronts or between the flow front and the wall of mold cavity [7, 10]. Their quantity and location are very important to be estimated. They also leave week spots in the product cross section like weld lined does. Additionally, this analysis helps to identify the possible locations of air vents to be provided in the mold cavity. The air trap result is shown in Fig. 7.

Figure 7. Air Traps

The result shows no air trapped in the cavity. The main reason behind this result can be the existence of parting line along length of mold cavity. This result eliminates the need of providing air vent in the mold. Still, as a standard practice it was decided to provide air vent for the cavities in the direction opposite to the injection location.

D. Frozen Layer Percentage at the Time of Ejection

The Frozen layer percentage at the time of ejection represents the percentage of thickness solidified at the end-of ejection process. The values of this result range from 0% to 100%. A greater value represents a thicker solidified layer and a lesser value represents thinner layer of frozen polymer [11]. It can be seen from Fig. 8 that majority of the cavity is solidified at the time of ejection. A minute portion near the junction of head and body is still solidifying. This is due to high material as well as heat accumulation in that region. There is no design solution possible for this. This issue can be handled by providing cooling channel in that region or by taking some time before opening the mold.

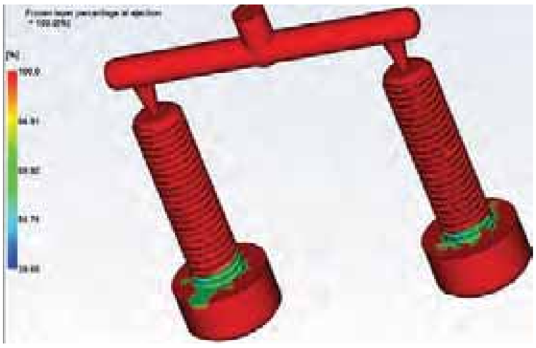

Figure 8. Frozen Layer Percentage at the Time of Ejection

Figure 9. Warpage Indicator

E. Warpage indicator

The warpage indicator shows the area of the part where the deflections may appear [12, 13]. Based on this result corrective actions can be taken to protect the correct profile of injection molded part. As can be seen from Fig. 9, that entire area is showing least possibility of warpage. Thus, there is no need to take any corrective measure into consideration.

F. Confidence of Fill

The confidence of fill result indicates the ease of filling the mold cavity i.e. the area of the mold may be difficult to fill with plastic [14]. The obtained result which is indicated in Fig. 10 is not very convincing as the entire threaded region is showing low confidence of getting filled. This issue can be tackled by increasing injection temperature and injection pressure at the time of mold filling. The process utilized 260°C injection temperature and 4 MPa injection pressure. It is not advisable to inject PP beyond 260°C for injection. Overheating can make plastic week and durability of plastic accumulated in threaded region will decrease drastically. Thus, it was decided to use higher injection pressure to ensure proper filling of the mold.

With the indications obtained from mold flow analysis, this work proceeded with manufacturing of model mold. Further the trial on the mold was taken which is discussed in next section.

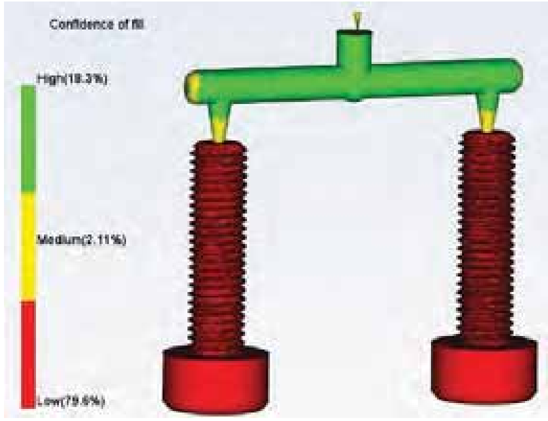

Figure 10. Confidence of fill

V. Die Trial

A die was prepared for trial purpose. This is unlike the actual injection molding die which is manufactured for industrial production. In Fig. 11 the manufactured die is shown. It is aligned with the nozzle of injection molding machine mentioned in Section III. The die was manufactured in three parts, according to split of parting line shown in Fig. 3. For better visibility of internal cavities one portion of the die is removed and shown in Fig. 11.

Figure 11. Mold for Trial

The machine was adjusted for values as per previous discussion. Injection pressure was considered 6 MPa which was beyond simulation value and less than the maximum capacity of machine. It was done to ensure proper filling as discussed in Section IV. The shot was taken to obtain the required impression, which is shown in Fig. 12. Multiple shots were taken to ensure the results obtained in the simulation. All the impressions obtained from HSHCS were observed against shrinkage, deflection, unfilled section etc. the obtained quality was found as per the results obtained in the simulation. In none of the shot unfilled section or any other defect was seen.

Figure 12. Obtained Impression (L) and Molded Plastic HSHCS (R)

VI. Conclusions

The choice of manufacturing processes is of strategic importance in new product development [1]. It has a significant impact on cost of production. The recent materials and manufacturing technology is making it possible to process any material by any manufacturing process. It is the responsibility of engineers to utilize technology for economic benefits. Scope this work is limited to evaluating feasibility of making HSHCS by injection molding process using plastic material. The content provided in this article can be utilized to design an injection molding die as well as to perform and analyze the mold flow analysis for plastic components. Modifying the die for industrial production, molding HSHCS by a type of polymer with comparable properties of a metal, testing these HSHCS in different loading conditions can be further scope of research work. This research attempted to provide researchers with an insight that there is always scope for betterment.

References

[1] David Kazmer, Amy M. Peterson, Davide Masato, Austin R. Colon, Joshua Krantz, "Strategic cost and sustainability analyses of injection molding and material extrusion additive manufacturing", Volume63, Issue3, March 2023, Pages 943-958, https://doi.org/10.1002/pen.26256

[2] Neeraj Kumar Jha, Bhavya Sri Tadiparthi, "*Design and Analysis of the Sleeve Ejection System in Injection Molding Die for Trolley Wheel*", CVRJST, Vol. - 17, pp. 132-137, 2019.

[3] E.Farotti, M.Natalini, "Injection molding. Influence of process parameters on mechanical properties of polypropylene polymer. A first study", Procedia Structural Integrity, Vol.- 8, 2018, pp. 256-264.

[4] R.Sánchez, A.Martinez, D.Mercado, A.CarbonelJ.Aisa, "*Rapid heating injection moulding: An experimental surface temperature study*", Polymer Testing, Volume 93, 2021, 106928

[5] Simion Haragas, Ovidiu Buiga, Ovidiu Buiga, Claudiu Popa, Florina Maria, ȘerdeanFlorina, Maria Șerdean," *Demolding Moment Calculation for Injected Parts with Internal Saw Thread*", MATERIALE PLASTICE, 2016, 53(3):378-381

[6] Giovanni Lucchetta, Davide Masato, Marco Sorgato, "Optimization of mold thermal control for minimum energy *consumption in injection molding of polypropylene parts*", Journal of Cleaner Production, Vol.- 182, 2018, pp. 217-226.

[7] Neeraj Kumar Jha, P V Ramana, "*Design Methodology and Analysis of Double Cavity Metal-Plastic-Insert Injection Molding Die for Push Board Pin*". CVRJST, Vol. - 14, pp. 91-96, 2018.

[8] N. Sateesh, S. Devakar Reddy, G. Praveen Kumar, Ram Subbiah," *Optimization of injection moulding process in manufacturing the flip cover of water meter*", Materials Today: Proceedings 26, 2020, pp. 240-246.

[9] Babur Ozcelik , Alper Ozbay, Erhan Demirbas, "*Influence of injection parameters and mold materials on mechanical properties of ABS in plastic injection molding*", International Communications in Heat and Mass Transfer, Vol.- 37, Issue 9, 2010, pp. 1359-1365.

[10] Abdulmecit Güldaş, Onuralp Uluer & Ahmet Özdemir, "*The Numerical Analysis of a Mold Cavity Filling Using the Finite Control Volume Method and Comparison to the Experimental Results*", Polymer-Plastics Technology and Engineering, 48:4, 2009 , pp. 389-396.

[11] Jiseon Hong, Sun Kyoung Kim, Young-Hak Cho, "*Flow and solidification of semi-crystalline polymer during micro-injection molding*", International Journal of Heat and Mass Transfer Vol.- 153, June 2020, 119576.

[12] Marton Huszar, Fawzi Belblidia, Helen M.Davies, Cris Arnold, David Bould, Johann Sien, "*Sustainable injection moulding: The impact of materials selection and gate location on part warpage and injection pressure*", Sustainable Materials and Technologies, Vol.- 5, 2015, pp. 1-8.

[13] Parth Patpatiya, Anshuman Shastri. "*Additive Manufacturing of Thermoplastic M8 Fasteners Using Photopolymer Jetting Technology for Low-Strength Fastening Applications*", 30 July 2021, PREPRINT (Version 1) available at Research Square [https://doi.org/10.21203/rs.3.rs-525671/v1]

[14] Czepiel M, Bańkosz M, Sobczak-Kupiec A. "*Advanced Injection Molding Methods: Review. Materials (Basel)*". 2023 Aug 24;16(17):5802. doi: 10.3390/ma16175802. PMID: 37687494; PMCID: PMC10489002.