



TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering

Vol. 68, Issue IV, November, 2025

DESIGN OF INJECTION-MOLDED PLASTIC PARTS, ACCORDING TO ISO STANDARDS

Cosmin-Gabriel GRĂDINARU, Gheorghe NAGÎȚ, Răzvan-Costel PRISECARU

Abstract: Currently, the trend among automobile manufacturers and automotive subsystem producers is rapidly shifting towards reducing manufacturing costs, driven by increasingly intense competition. One of the solutions to achieve this is replacing metal materials with plastics, which offer lower costs and a more efficient mass-production process. This paper presents part of the design methodology for an injection-molded plastic part by calculating the precision grade and applying tolerances in accordance with ISO 20457:2018. Additionally, it introduces a tool designed to facilitate the designer's work in this area. This tool leverages information from the standard, automating the necessary steps for implementation while providing supplementary explanations and examples that extend the standard's information.

Keywords: plastic molded workpiece, geometric dimensioning and tolerancing (GD&T), precision grade.

1. INTRODUCTION

Plastic injection molding represents an important component of mechanical engineering, being one of the most efficient and precise manufacturing methods used at an industrial scale. This process allows the efficient and repeatable production of components with complex geometries, optimizing material usage and reducing costs. Strict control of key parameters—such as temperature, pressure, and cooling time—ensures superior mechanical properties and increases the quality of the final products.

The automotive industry increases the need for plastic components because of their advantages in reducing product weight, improving fuel consumption, and lowering expenses. Complex materials facilitate all types of efficient fabrication of parts, minimize the need for later processing, and maintain safety and durability requirements, providing also a viable alternative to metallic materials.

This process involves heating the material until it reaches a plastic state, followed by injecting it under pressure into the mold. Once the mold cavity is filled, the material cools and solidifies, taking the desired geometry. Then the mold opens to eject the part, and the

manufacturing cycle is repeated, ensuring an efficient production process [15].

This field shows significant international interest, leading numerous authors to explore and publish research on various related topics. Danu et al. investigated the processing of polymers through extrusion and injection molding [7]; Oktem et al. conducted a study on the design, analysis, and manufacturing of plastic parts incorporating biopolymer-based materials, such as polypropylene (PP) and acrylonitrile butadiene styrene (ABS) [18]; Sánchez-Soto et al. proposed optimization methods for gas-assisted plastic injection molding, focusing on an automotive plastic cover [21]; Zhao et al. examined the plastic part manufacturing process using FEM analysis on different structural configurations [27]; Zabala et al. described the types, causes, and prevention strategies for injection mold wear [26]; Barbosa et al. defined a method for designing injection molds for plastic components [4]; Almeida et al. developed a design strategy for movable mechanisms used in injection molds [1]; Asanuma et al. presented a method for developing the plastic waste injection process using blast furnace systems [3]; Hopmann et al. studied a method for minimizing material usage in the manufacturing of plastic parts [16]. All these papers address different topics in the field

of plastic injection, but the method of creating the execution drawing for a plastic part is not present, which constitutes the motivation for the research topic.

In the field of patents, Schar et al. proposed a method for producing molded parts by injection molding [22]; Altonen et al. described a system for feeding raw materials into the injection process [2]; Benyahia et al. patented a method for manufacturing plastic parts for vehicles [5]; Deckert et al. developed a component and its corresponding manufacturing method [8]; and Franksson et al. introduced a method for fabricating plastic parts [10].

When designing a plastic part, it is essential to follow the principles and methodologies from the specialized literature. Among the requirements for creating the 3D model are maintaining a uniform wall thickness, determining the draft angle based on the material used (influenced by the friction coefficient), complying with the minimum wall thickness requirements, and correctly defining the parting line to avoid sharp edges. Additionally, shut-off angles must be configured to minimize friction between mold components, and the ejection system should be optimized to reduce the number of tool parts, lowering the costs, because the manufacturing of the mold is one of the main aspects regarding the costs [6], [9], [19].

Regarding technical drawings, compliance with current standards is essential. ISO 16792:2021 [25], ISO 14405-1:2015 [12], and ISO 14405-2:2018 [13] define general requirements for graphical representation and the application of tolerances to the parts. Moreover, specific standards apply to different types of parts, such as ISO 6930-2:2011 for components produced by cold plastic deformation (stamping), ISO 965-2:1999 [17] for screws and nuts, ISO 8062-3:2007 [11] for parts produced through metal injection molding, and ISO 20457:2018 [20] for components manufactured via plastic injection molding, which regulate applicable tolerance values and specifications.

This paper presents a methodology for applying current standards in the design of injection-molded plastic parts, focusing on determining the required precision grade and the application of dimensional and geometrical

tolerances to functional surfaces and general tolerances. To facilitate this, a Microsoft Excel-based tool was developed to optimize the process of defining, automating, and documenting the necessary steps, thereby facilitating the analysis and implementation of technical requirements in the design process. In addition to referencing standard-specific details, the tool provides illustrative examples and supplemental explanations, featuring an intuitive and user-friendly interface for ease of operation.

A proposed workflow chart for a generic plastic part is shown below, in Fig. 1. It presents the main steps that a part could get through, from the order intake to documentation and delivery to the customer. The drawing execution is highlighted being the focus of this research.

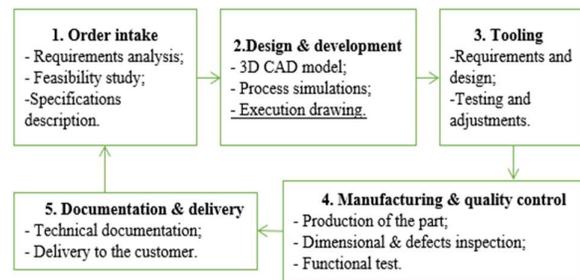


Fig. 1. Proposed workflow chart for a plastic part.

2. METHODOLOGY

To accurately finalize a technical drawing for a part produced through plastic injection molding, in accordance with ISO 20457:2018, a series of well-defined steps must be followed, as illustrated in the logic diagram shown in Fig. 2.

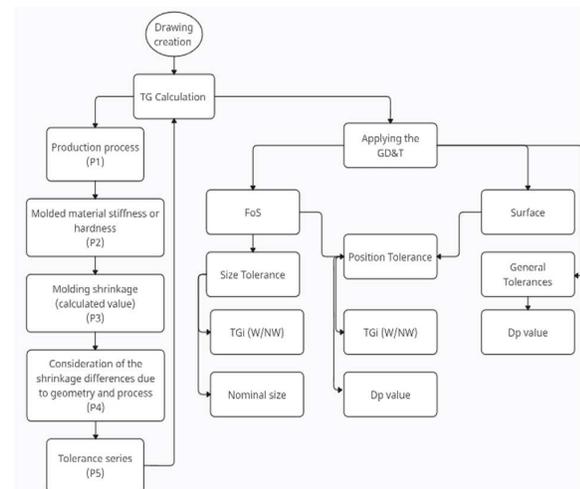


Fig. 2. Logic diagram of methodology.

In the initial phase, the tolerance grade of the part—denoted as parameter Tg —is calculated. This parameter is defined as the sum of several contributing factors (or points) which are defined in the standard.

Accordingly, parameter $P1$ is taken directly from the standard, depending on the type of manufacturing process; $P2$ is determined by the material of the part; the final three points are established in close collaboration with the part supplier. Specifically, parameter $P3$ is a function of the calculated value of the part's shrinkage (VS), as defined by Equation (1) below.

$$VS = \left(1 - \frac{L_f}{L_w}\right) \cdot 100 [\%] \quad (1)$$

where L_f is the molded part dimension (measured after 16 h to 24 h after production) and L_w is the tool contour dimension (at $23^\circ\text{C} \pm 2^\circ$). These calculations are performed for multiple dimensions across several parts, to obtain a statistical value with minimal error.

Parameter $P4$ depends on the statistical accuracy in determining the part's shrinkage—based on prior experience, systematic measurements, or simulations—while parameter $P5$ is determined by the type of production, in terms of required precision and quality control rigor.

Furthermore, the standard defines three categories of tolerances: size tolerances (applicable only to features of size), position tolerances, and general profile tolerances (general tolerances). The values for these tolerances depend on the range in which the nominal dimension falls. To correctly determine the size and position tolerances, in addition to the tolerance grade, it is necessary to identify if the dimension is tool-specific (W) or non-tool-specific (NW)—that means if the feature is generated by a single mold component (W) or by multiple components (NW), as shown in Fig. 4.

Size tolerances are applied exclusively to features of size (FoS), which, according to the standard, may include cylindrical, spherical, conical surfaces, two opposing parallel planes, a circle, and so on.

Position tolerances, on the other hand, may be applied to any feature, regardless of if it's qualifying as a FoS or simply a surface.

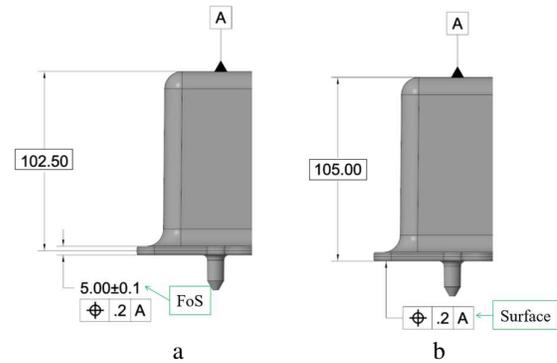


Fig. 3. Example of a FoS and a surface tolerancing (extracted from the tool).

Fig. 3a illustrates the tolerancing method for both a feature of size (FoS) and Fig. 3b for a simple surface. The distinction is based in how the part is considered for tolerancing—specifically, which surface is functionally critical.

Any FoS leads to the extraction of a median plane, which is then assigned a position tolerance of 0.2 mm, as shown in Fig. 3a. However, suppose the functionally important surface is the one used for contact with other parts (e.g., the resting surface of the flange onto a lower part). In that case, it is recommended to tolerate that surface, allowing for precise control of its compliance within the 0.2 mm position tolerance.

In this case, the upper surface of the flange is not tolerated as a FoS and instead receives either an explicit individual tolerance or it is constrained by the general tolerances specified on the drawing.

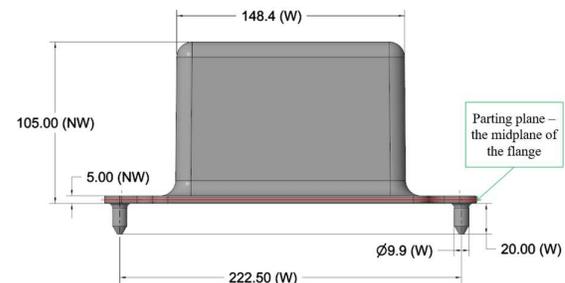


Fig. 4. Examples of W and NW dimensions (extracted from the tool)

General profile tolerances are indicated on the technical drawing, in accordance with ISO 22081:2021 [14], either within or near the title block, these are no longer regarding of Tg .

3. EXAMPLE OF APPLYING THE METHODOLOGY USING THE EXCEL TOOL

To illustrate the process as clearly as possible, the method is applied to a predefined model. This model is a cover-type part, made of PBT GF 30 material. The draft angle is set at 1° according to references [6], [9], [19].

As previously mentioned, the first step was to calculate the part’s precision class, specifically the T_g parameter, which resulted in a value of 5 (Fig. 5).

| Drop down lists | | | |
|-----------------|--|---|------|
| Par. | Category | Pick variables | Val. |
| P ₁ | Production process | Injection moulding / injection compression moulding / transfer | 1 |
| P ₂ | Moulded material stiffness | Modulus of elasticity above 1200 N/mm ² & Shore D above 75 | 1 |
| P ₃ | Moulding shrinkage | above 1% to 2% | 2 |
| P ₄ | Consideration of the shrinkage differences due to geometry and process | Precisely possible with limitations - deviations are max. ±20 % | 2 |
| P ₅ | Tolerance series | Series 2 (accurate production) | -1 |
| RESULTS | | | |
| Type | Formula | Value | |
| P _g | P ₁ +P ₂ +P ₃ +P ₄ +P ₅ | 5 | |
| T _g | Table 3 | 5 | |

Fig. 5. The calculation of the T_g by the tool.

In Fig. 5, using the drop-down lists in the Excel tool (light green cells), values for the P_i parameters were selected according to the standard. In the blue cells, the tool displays results according to the operator’s selections.

Thus, P₁ resulted in a value of 1 due to the manufacturing technology used; P₂ was assigned a value of 2 because the material’s modulus of elasticity is 3400 N/mm² [23]; for P₃, the value of 2 resulted from the calculation using Equation (3); the values for P₄ and P₅ were assigned arbitrarily, assuming a discussion with the part supplier.

After selecting all parameter values, the tool automatically returns their sum and the corresponding precision grade according to the standard.

Next, the technical drawing of the part can be created using the automated tool to assign the tolerance values. An example of a draft (incomplete, without surface quality requirements, tests, etc.) can be seen below, in Fig. 9. To explain the methodology there are three examples, from Fig. 6 to 8, to be discussed.

Example #1 – the tool allows the selection from the drop-down list (light green cells from Fig. 6) of the previously calculated tolerance grade 5 of type W (since it is the result of a single injection mold component). Thus, for the height of positioning pins, which is 20 mm, falling within the selected range of (18-30) [mm], the tolerance value of ±0.17 mm results (blue cell). Additionally, from the corresponding table of all size tolerances, it will highlight in dark green the row and column corresponding to the tolerance and create a black border around the cell.

In the industrial environment, tolerance values like these are avoided unless there is a strict requirement. After a discussion with the supplier, the value could be rounded to ±0.2 mm.

| Tolerance Grade | Size interval | Tolerance value |
|-----------------|---------------|-----------------|
| TG5 - W | >18 to 30 | ±0,17 |

Fig. 6. The result from the Excel tool for example #1.

Example #2 – for selecting the value of the position tolerance, the tool operates in the same way, by inserting in the drop-down list the tolerance grade TG5 – NW (since the upper surface of the cover is generated from a different mold component than Datum A) and the corresponding range for the distance D_p (which is the maximum distance from the part's origin, i.e., the intersection of Datum A and Datum B, to the nominally tolerated feature), which returns a value of 1.4 mm, as shown in Fig. 7.

| Tolerance Grade | D _p Size Interval | Tolerance value |
|-----------------|------------------------------|-----------------|
| TG5 - NW | >80 to 120 | ∅ 1,40 |

Fig. 7. The result from the Excel tool for example #2.

Example #3 – The same method was applied, resulting in a position tolerance of 2.18 mm, presented in Fig. 8 (2.2 mm as indicated on the drawing) for the position of all four clearance holes for fixation. The distance D_p is considered as the largest distance from the origin to the furthest through hole (the diagonally opposite one), resulting from the combination of the two theoretically exact (base) dimensions.

| Tolerance Grade | D_p Size Interval | Tolerance value |
|-----------------|---------------------|-----------------|
| TG5 - NW | >315 to 400 | $\phi 2,18$ |

Fig. 8. The result from the Excel tool for example #3.

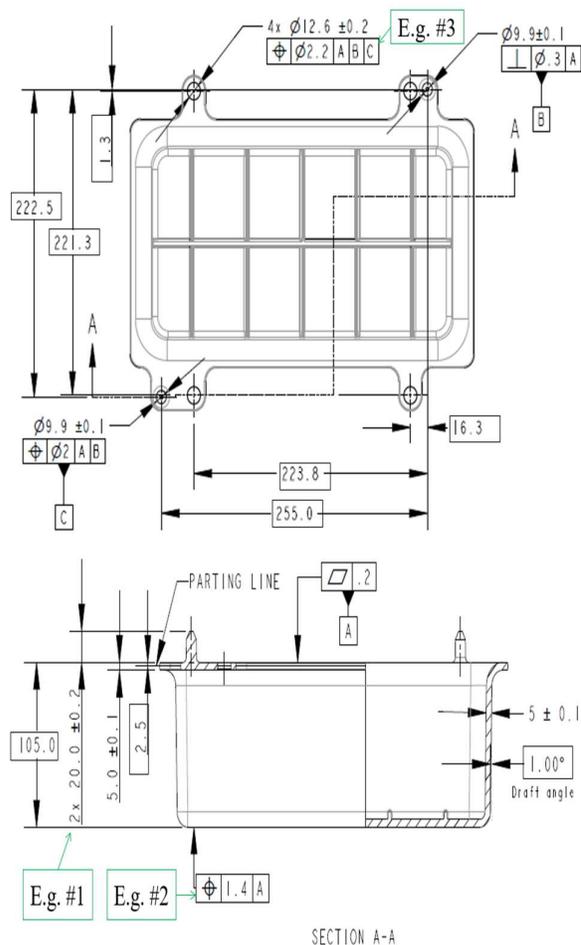


Fig. 9. Example of a draft drawing using the research methodology.

General profile tolerances no longer consider the precision grade or the method of feature generation, but only the range in which the corresponding D_p value is found.

Therefore, since these are variable general tolerances, it is necessary to include them in a table on the drawing, in accordance with ISO 22081:2021.

The applicable tolerances according to the standard can be changed, but only after a discussion with the part supplier and their approval. Sometimes, the supplier may request a relaxation of the tolerances for technological capabilities reasons.

4. CONCLUSION

The design of a part manufactured by plastic injection molding requires specialized knowledge and increased attention, as it is necessary to respect a relatively high number of rules, both for 3D modeling and for technical drawing.

Such parts are increasingly used instead of metal ones due to considerations of cost, technological feasibility, and productivity, both in the automotive industry and in the wide industrial sector.

ISO 20457:2018 standard specifies the methodology for creating the technical drawing by calculating the tolerance grade, which influences the value of the applied tolerances, and, of course, the precision and cost of the final part.

In the future, the field will be explored in more detail, focusing on the correct creation of the 3D model, as well as a potential cost analysis to highlight the influence of a higher or lower tolerance grade compared to the one calculated for a part.

5. REFERENCES

- [1] Almeida F., Sousa V., Silva F., Campilho R., Ferreira L., *Development of a novel design strategy for moving mechanisms used in multi-material plastic injection molds*, Applied Sciences, DOI: 2076-3417/11/24/11805, 2021.
- [2] Altonen G., Berg C., Neufarth R., Schiller G., *Non-naturally balanced feed system for an injection molding apparatus*, patent US9937656 B2.
- [3] Asanuma M., Ariyama T., Sato M., Murai R., Nonaka T., Okochi I., Tsukiji H., Nemoto K., *Development of waste plastics injection process in Blast furnace*, ISIJ International, Vol. 40, p. 244-251, 2000.
- [4] Barbosa R., Campilho R., Silva F., *Injection mold design for a plastic component with blowing agent*, Procedia Manufacturing, Vol. 17, p. 774-782, 2018.
- [5] Benyahia R., Lux T., Turan A., Fournier N., *Method for producing a plastic vehicle part*, patent US 10603826 B2.
- [6] Clive M., *Design guides for plastics*, Tandram Econology Ltd, 2021, <https://tangram.co.uk/>.
- [7] Danu M., Ibănescu C., Ibănescu A., *Prelucrarea polimerilor prin extrudare si injectie*, Iași, <http://www.didactic.icpm.tuiasi.ro/cv/ibanescuconstanta/cursuri%20web/PPEI.pdf>.
- [8] Deckert K., Hahn M., *Plastic moulded part and method for production thereof*, patent US10875222 B2.

- [9] Eurotec, *Part Design Guideline*, <https://www.eurotec-ep.com/en>.
- [10] Franksson O., Axelsson R., *Method for injection molding plastic parts by an injection molding machine*, patent US10279562 B2.
- [11] *General dimensional and geometrical tolerances and machining allowances for castings*, standard ISO 8062-3:2007.
- [12] *Geometrical product specification (GPS) – Dimensional tolerancing – part 1: linear sizes*, standard ISO 14405-1:2016.
- [13] *Geometrical product specification (GPS) – Dimensional tolerancing – part 2: dimensions other than linear or angular sizes*, standard ISO 14405-2:2018.
- [14] *Geometrical product specifications (GPS) – Geometrical tolerancing – General geometrical specification and general size specification*, standard ISO 22081:2021.
- [15] Hargas S., *Proiectarea matrițelor de injectat*, Technical University of Cluj-Napoca, 2004.
- [16] Hopmann Ch., Neub A., Weber M., Fischer T., *Plasticizing and injecting lowest quantities of plastic*, AIP Conference, DOI: 10.1063/1.4873753, 2014.
- [17] *Limit of sizes for general purpose external and internal screw threads – medium quality*, standard 965-2:1999
- [18] Oktem H., Karasungur H., Oz A., *Design, analysis and manufacturing of plastic parts made from bio-added polymers*, EPSTEM, ISSN:2602-3199, Vol. 32, P. 238-243, 2024.
- [19] Opran C., *Tehnologii de injecție în matrițe*, Polytechnic University of Bucharest, 2013.
- [20] *Plastics moulded parts – tolerances and acceptance conditions*, standard ISO 20457:2018.
- [21] Sanchez-Soto M., Gordillo A., Arasanz B., Aurrekoetxea J., Aretxabaleta L., *Optimizing the gas-injection moulding of an automobile plastic cover using an experimental design procedure*, Journal of Materials Processing Technology, Vol. 178, p. 369-378, 2006.
- [22] Schar M., Zurfluh P., *Method for producing brushes, in particular interdental brushes, and brush, in particular interdental brush, and product group comprising a plurality of brushes*, patent US2019/0246778 A1.
- [23] Schmit & Bartl, *PBT GF 30 – material data sheet*, https://www.schmidt-bartl.de/pdfs_en/halbzeuge/technischdatenblaetter/PBT_GF30_en.pdf.
- [24] *Stamped steel parts – general tolerances*, standard ISO 6930-2:2011.
- [25] *Technical product documentation – digital product definition data practices*, standard ISO 16792:2021.
- [26] Zabala, B., Fernandez, X., Rodriguez, J.C., López-Ortega, A., Fuentes, E., Bayón, R., Igartua, A., Girot, F., *Mechanism-based wear models for plastic injection moulds*, Elsevier, ISSN 0043-1648, 2019.
- [27] Zhao M., Tang Z., *Finite element simulation of injection mold design integrating different structures of conformal cooling channels*, Processes, Vol. 13, DOI: 2227-9717/13/1/234, 2025.

Proiectarea pieselor din plastic realizate prin injecție, în conformitate cu normele ISO.

Rezumat. În prezent, tendința predominantă în rândul producătorilor din industria auto și al furnizorilor de subsisteme auto este orientată tot mai accentuat spre reducerea costurilor de fabricație, ca răspuns la intensificarea competiției pe piață. Una dintre soluțiile identificate pentru atingerea acestui obiectiv constă în înlocuirea materialelor metalice cu materiale plastice, acestea oferind avantaje precum costuri mai reduse și un proces de producție în masă mai eficient. Lucrarea de față prezintă o parte a metodologiei de proiectare a unei piese realizate prin injecție din plastic, prin calcularea gradului de precizie și aplicarea de toleranțelor geometrice și dimensionale, în conformitate cu standardul ISO 20457:2018. De asemenea, este introdus un instrument dedicat sprijinirii activității proiectantului în acest domeniu. Acest instrument utilizează informațiile din standard, automatizând pașii necesari implementării și oferind explicații și exemple suplimentare, care depășesc conținutul explicit al standardului.

Cosmin-Gabriel GRĂDINARU, Student, „Gheorghe Asachi” Technical University of Iași, Department of Machine Manufacturing, cosmin-gabriel.gradinaru@student.tuiasi.ro.

Gheorghe NAGÎȚ, Professor, „Gheorghe Asachi” Technical University of Iași, Department of Machine Manufacturing, gheorghe.nagit@academic.tuiasi.ro.

Răzvan-Costel PRISECARU, Senior Mechanical Engineer – Tolerance Calculation, Schaeffler Romania S.R.L., razvanprisecaru97@gmail.com.