

Article

Cross-Study of Techniques for the Analysis of Deformations Generated in the Injection Molding Process

Vladimir Zagoya-Juárez ¹, Héctor Plascencia-Mora ^{2,*}, Jaime Navarrete Damián ³ , Ismael Ruiz-López ⁴ ,
Juan Francisco Reveles Arredondo ²  and María Cristina López-Mendez ^{1,*} 

¹ División de Estudios de Posgrado e Investigación, Tecnológico Nacional de México, Instituto Tecnológico Superior de Misantla, Km 1.8 Road to Loma del Cojolite, Misantla 93821, Veracruz, Mexico; vzagoyaj@itsm.edu.mx

² Departamento Ingeniería Mecánica, DICIS, Universidad de Guanajuato, Carr. Salamanca-Valle 3.5+1.8 km, Palo Blanco, Salamanca 36885, Guanajuato, Mexico; fco.reveles@ugto.mx

³ Departamento de Diseño y Desarrollo de Equipo, Tecnológico Nacional de México, CRODE, Celaya 38020, Guanajuato, Mexico; jaime.navarrete@crodecelaya.edu.mx

⁴ Pintura Estampado y Montaje S.A.P.I. de C.V., Avenida Concepción Beistegui 2007 Col. Zona Industrial Poniente, Celaya 38020, Guanajuato, Mexico; ismael.ruiz@cieautomotive.mx

* Correspondence: hplascencia@ugto.mx (H.P.-M.); mclopezm@itsm.edu.mx (M.C.L.-M.)

Abstract

Injection molding is a plastic material processing technique used in the polymer industry. Because it is a complex process that requires injection cycles to achieve the desired aesthetic quality in the molded parts, it is essential to evaluate and configure all process parameters to predict and reduce defects, thereby decreasing the processing time and energy consumption. This study presents the results of tests performed on molded HDPE parts, including modeling and simulation using ANSYS® (2025 R1), a design of experiments (DOE), and 3D scanning of the molded parts. The study compares the behavior of defects (warpages and sink-marks) in molded parts using 3D scanning with the results obtained from coupled thermal-structural field finite element simulations. These simulations were performed using software to assess the residual thermal stress of the ejection phase. The results visually display information that helps designers and engineers in the polymer processing sector evaluate molding-process failures using different software.

Keywords: structural-thermal simulation; warpage; sink-mark; Scan 3D; DOE; injection molding

1. Introduction

The plastic injection molding industry is characterized by its ability to generate high-volume, rapid, and cost-effective mass production. These characteristics allow for the creation of parts with complex geometries using thermoplastic and thermoset materials for various applications, including everyday items, automotive parts, and healthcare products. The plastic injection molding process consists of phases well-known in the industry, which can be grouped as follows: feeding and heating the raw material until it melts within the barrel; mold clamping; injection, and filling with the molten plastic under pressure; cooling to solidify the part; and mold opening for the ejection of the formed part. However, this type of process is not without its flaws, including defects such as flash, burn marks, warpage, sink-marks, and shrinkage cavities, all of which can affect the quality of the final product.



Academic Editor: Guido Di Bella

Received: 29 July 2026

Revised: 31 August 2026

Accepted: 1 September 2026

Published: 3 September 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

Specifically, this study focuses on two types of defects that occur in the formed part—analyzing them during the ejection phase—namely warpage, one of the most critical failures, which can be characterized linearly or volumetrically and is linked to compressibility; and sink-mark, which is surface deformation caused by non-proportional volumetric contraction in areas with greater thermal mass during the holding and cooling phase [1,2].

To eliminate these types of failures, the plastic injection molding industry is gradually promoting the demand for professionals who understand and can calculate production costs, select the appropriate plastic material for the part's functionality, model high-quality parts with complex geometries and tight tolerances, predict the part's resistance to stress, and, above all, adjust the process to reduce waste or non-conforming product rates.

Globally, the most widely used materials are polypropylene (PP) and high-density polyethylene (HDPE). Both stand out for their versatility, good technical performance, and low cost, making them fundamental pillars of industrial production using plastic injection molding [3].

This research focused on high-density polyethylene (HDPE), a tough thermoplastic known in industry to be prone to shrinkage failure, which can lead to indentations in thicker areas of the part. This is because the material's molding parameters are strongly influenced by the Pressure, Volume, and Temperature (PVT) history and process conditions [4].

Also, this study proposes that computer-aided engineering (CAE) software used as computational tools can be highly effective in generating predictive models to visualize potential failures in the thermoplastic injection molding process, as these programs often solve models using computational fluid dynamics (CFD) [5]. Consequently, for a more rigorous analysis and experimental validation of the three-dimensional deviations obtained from the parts generated by injection molding [6], this work proposes the integration of non-conventional finite element analysis (FEA) techniques, such as the results obtained from solidification using thermal-structural analysis with the software ANSYS®; the comparison of parts scanned in three dimensions (3D) versus the predicted part generated by Computer-Aided Design (CAD) to generate deviation maps using the Software GOM Inspect 2019®, providing information that can overcome limitations of purely commercial software approaches and traditional empirical validation.

2. Materials and Methods

2.1. Overview of the Experiment

The experimental design and its execution were adapted to meet the production requirements of parts as demanded by the plastic injection molding industry. The basic geometry of the tensile test specimens was used, following the ASTM D638 (Standard Test Method for Tensile Properties of Plastics. ASTM International: West Conshohocken, United States, 2022) standard, as shown in Figure 1. The parts were injections molded from HDPE because, as a thermoplastic characterized by being partially amorphous and partially crystalline, it can exhibit injection molding shrinkage anisotropy, meaning different properties depending on the direction in which it is measured [7]. This is due to the orientation of the plastic molecules during the injection molding process, which can affect the strength and ductility of the material, which directly impacts the generation of warpage [8].

To perform the Design of Experiments (DOE), the variation of the injection parameters recommended by the material manufacturer and by the recommendations of the specialist personnel in the plastic injection molding process were considered; these parameters were: mold temperature, injection temperature, pressure, and holding time.

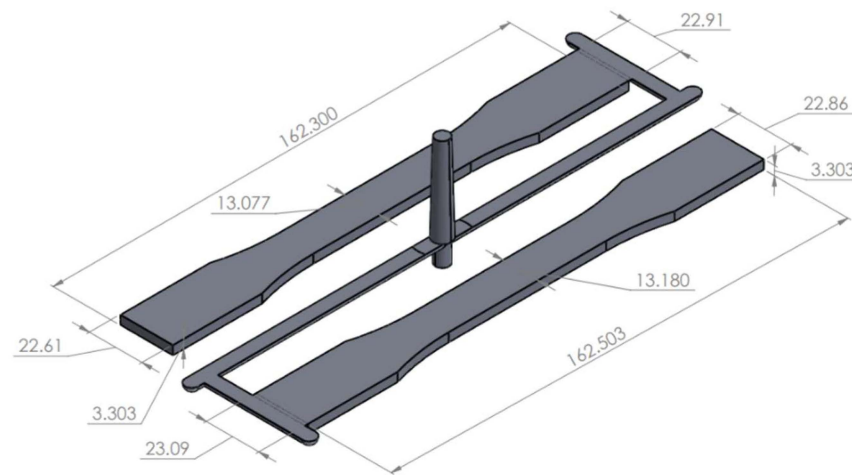


Figure 1. Geometry model of the injection mold cavity of specimens according to ASTM D638.

To evaluate the specific impact of individual process conditions, their levels, and mutual combinations on the dimensional stability of the produced sample, the statistical method of Design of Experiments was applied as shown in Table 1.

Table 1. Considered input factors and their levels.

Input Factor	Process Parameter	Test Level (Min. Value)	Test Level (Max. Value)	Unit
A	Inject Temp	200	220	°C
B	Holding Time	1.5	4.5	s
C	Mold Temp	7.5	17.5	°C
D	Hold Press	390	510	Bar

Once the parameters were defined, the test samples were taken, and then FEA software simulations were performed with the software ANSYS® (2025 R1) to simulate the cooling and deformation of the part once it was ejected from the mold. Based on the results, a more detailed understanding of the physical processes during the injection molding cycle was achieved.

Similarly, a sample of the ejected parts was classified according to the parameters defined by the DOE, a 3D optical scan was performed, and then a comparison was made with the desired geometries generated with the CAD model. Since this comparison method has proven to be very effective in determining displacements and deformations [9], these differences were then refined and processed to obtain statistical data showing the percentages of deformation and shrinkage at different reference points. Figure 2 shows a schematic plan of the experiment.

To perform the analysis, parameter ranges such as melting temperature, mold temperature, pressure, and holding time were defined and have been considered as the main parameters that influence the main factors for the generation of Warpage [10] and are taken as a starting point to generate the DOE as follows:

- Melting temperature: 200–220 °C, as a starting point (excessive temperatures increase the risk of warpage/sink-mark).
- Mold temperature: 20–60 °C (typically), increases crystallization and warpage in HDPE.
- Holding pressure: determined to study its effect on warpage/sink-mark, from lowest to highest pressure.

- Holding time: determined to study its effect on warpage/sink-mark, from shortest to longest time.

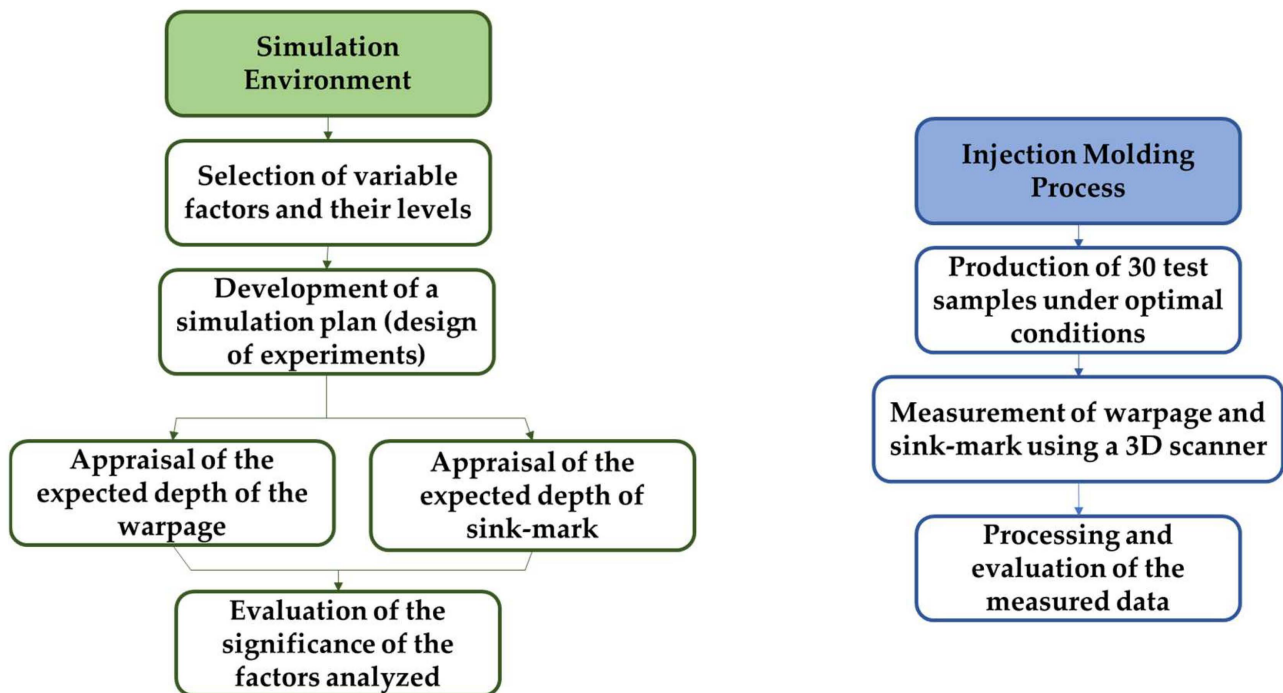


Figure 2. Schematic plan of the experiment.

The defined DOE is a composite central design, as can be seen in Table 2, which is a response surface methodology, and the factor levels were first determined by preliminary tests with parameters to obtain a part of acceptable quality, once parts of acceptable quality were obtained, and from those levels, the factor levels were varied. Once the parameter limits were established, the DOE was generated, resulting in the following:

Table 2. DOE used for study.

Run	Factor 1 A: Inject Temp °C	Factor 2 B: Holding Time Sec	Factor 3 C: Mold Temp °C	Factor 4 D: Hold Press Bar
1	205	4.5	10	420
2	205	4.5	15	420
3	205	4.5	10	480
4	205	4.5	15	480
5	210	3.5	12.5	450
6	210	2.5	12.5	390
7	215	4.5	10	480
8	205	2.5	15	420
9	210	3.5	12.5	450
10	215	2.5	10	480
11	215	4.5	15	480
12	215	2.5	15	480
13	210	3.5	12.5	480

Table 2. Cont.

Run	Factor 1 A: Inject Temp °C	Factor 2 B: Holding Time Sec	Factor 3 C: Mold Temp °C	Factor 4 D: Hold Press Bar
14	215	4.5	10	420
15	215	2.5	15	420
16	205	3.5	10	480
17	205	2.5	10	420
18	210	3.5	12.5	510
19	210	5.5	12.5	450
20	210	3.5	17.5	450
21	215	2.5	10	420
22	210	1.5	12.5	450
23	220	3.5	12.5	450
24	210	3.5	12.5	450
25	200	3.5	12.5	450
26	215	4.5	15	420
27	210	3.5	12.5	450
28	210	3.5	12.5	450
29	210	3.5	7.5	450
30	205	2.5	15	480

2.2. Materials

The materials used in this study included a commercial-grade high-density polyethylene (HDPE) homopolymer from LyondellBasell Alathon H5220 produced in Texas, United States, Injection Molding Grade, which provides easy processing characteristics and exhibits excellent toughness properties and color as well as low odor and good processing stability. Typical applications include housewares, containers, caps, and closures. The density of the polymer was 0.952 g/cc, melt flow 20 g/10 min, load 0.980 Kg, temperature 190 °C, melting point 130.2 °C, tensile modulus (elastic), 1260 MPa (ASTM D638, Standard Test Method for Tensile Properties of Plastics. ASTM International: West Conshohocken, United States, 2022), flexural modulus 1250 MPa (ASTM D790, Standard Test Methods for Flexural Properties of Unreinforced and Reinforced Plastics and Electrical Insulating Materials. ASTM International: West Conshohocken, United States, 2017), and isotropic secant coefficient of thermal expansion $1.45 \times 10^{-4}/^{\circ}\text{C}$, and these physical properties and process parameters were compiled from the technical data sheet provided by the material supplier and were used for the thermo-structural analysis in ANSYS® software.

2.3. Injection Molding Machine

The molded parts were formed using an injection molding machine (WITTMANN BATTENFELD EcoPower 110, Wittmann Battenfeld GmbH, Kottlingbrunn, Austria). This machine is shown in Figure 3a, with a clamping force of 1100 kN, with the following dimensions: length/width/height: $4.50 \times 1.50 \times 2.10$ m and weight of 4800 kg. Similarly, Figure 3b shows an image of how the machine's injection parameters are set.

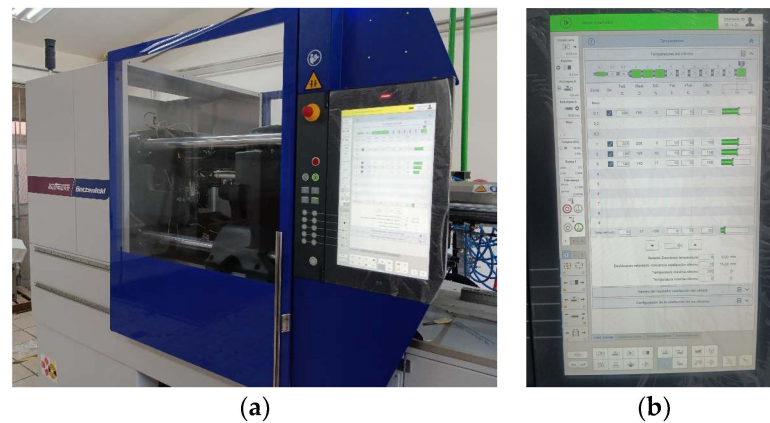


Figure 3. Injection molding machine WITTMANN BATTENFELD EcoPower 110.

2.4. Finite Element Model

While injection molding analysis is based on flow, heat transfer, and solid mechanics models, the term non-conventional FEA software refers to the application of advanced simulation modules and methodologies that go beyond standard fill/compaction/cooling analysis. Unlike simplified shrinkage models, thermo-structural analysis, often integrated into FEA platforms, computes residual stress (containing shrinkage, cavitation, and buckling) induced by temperature and pressure gradients as the material transitions from a molten to a solid state. Unconventional FEA software tools such as ANSYS® can provide a much more accurate approximation of the phenomena presented during all phases of the injection process and simulate the structural deformations that may occur [11,12]. Furthermore, an advantage found when modeling a plastic injection process using fluid-structural interaction (FSI) analysis is that this analysis allows us to export temperature locations to structural analyses, which can be applied in the mold design process, used to optimize the process solidification phase; since there are results in thermo-structural simulation studies versus experimental results generated with the ANSYS® Workbench modules with differences of 1.93% to 5.8%, as mentioned by Páramo et al. [13], this gives certainty that the thermo-structural results generated with the software resemble experimental results. For this study, the simulation of the CAD part was defined by considering the physical phenomena occurring during the transient solidification phase after the part is ejected from the mold. Table 3 shows the simulation parameters of the thermostatic mesh process.

Table 3. FEM simulation parameters of Mesh Thermal-Static process.

FVM simulation parameters ANSYS	
Mesh type	Multizone
Physics Preference	Explicit
Element size	1×10^{-3} m
Nodes	64,903
Elements Volume	13,000
Element Order	Quadratic

To ensure the analysis results are accurate, finite element analysis (FEA) will play an important role in the simulation. For this purpose, the square shape of the surface mesh was chosen for the finite element analysis as shown in Figure 4a. To simulate the phenomenon of the transient solidification phase once the piece was ejected from the mold, the Ansys® Workbench (finite element analysis, or FEA) was used for the thermal-structural simulation, as it is a tool suitable for capturing local pressure and temperature gradients, as

well as deformation effects resulting from the transient solidification phase, employing the following systems:

- **ANSYS Transient Thermal Analysis:** It serves to determine the variation in temperature distribution over time—factors of interest regarding the transient solidification phase—and, for this study, the temperatures obtained through transient thermal analysis are used as input data for the structural analysis aimed at evaluating the resulting thermal stress.
- **ANSYS Static Structural:** To evaluate the deformation induced by thermal volumetric contraction.

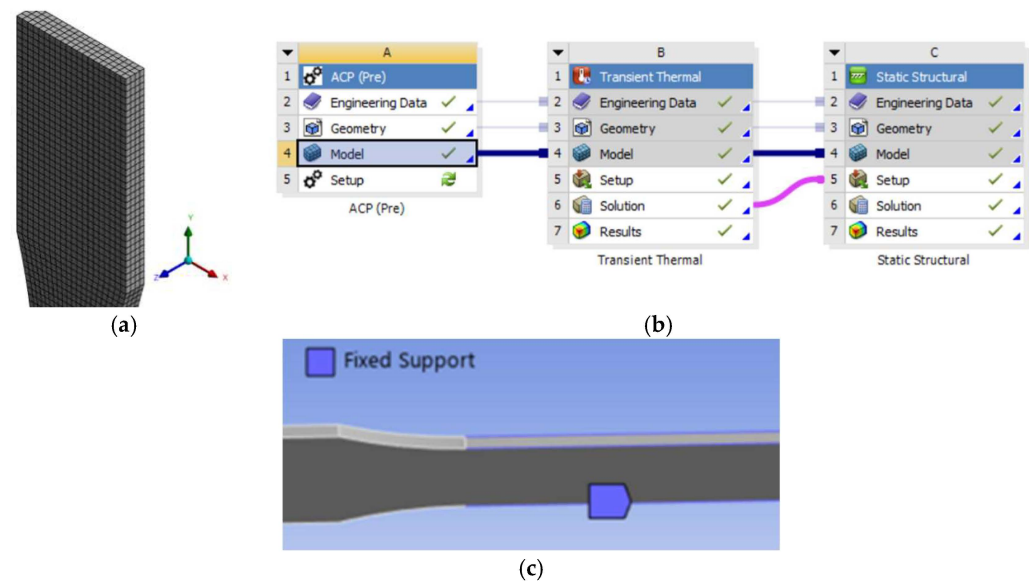


Figure 4. (a) Finite element analysis of the finalized part, (b) thermo-structural simulation structure interaction model for the simulation process in Ansys® Workbench, and (c) edge with fixed support along all four sides.

The analysis was performed using ambient thermal condition parameters for the post-injection simulation of the test specimens during the transient solidification process to ambient temperature at 25 °C, natural convection coefficient at 12 W/m² °C, and the temperature of the molded part was determined using values obtained from Equation 4, which is presented later in this document to establish different temperatures based on parameters defined in the DOE, as seen in Figure 4b and in Figure seen edge with fixed support along all four sides. The material's mechanical parameters defined for the thermo-structural simulation of the solidification phase were established in Section 2.3; additionally, Figure 4c shows the fixed support conditions applied to the four edges defined in the model for the static structural analysis.

2.5. Ejection Temperature Calculations

Polymer injection molding is a rapid, high-pressure process in which cooling plays a crucial role in determining the dimensions of the final product. For semi-crystalline polymers, the degree of crystallinity depends on the cooling rate to ensure proper solidification. Furthermore, semi-crystalline polymers exhibit more pronounced phase transition regions than amorphous materials, as their density increases with crystallization. An increase in pressure raises the temperature at which crystallization begins. Ultimately, HDPE crystallizes between 50 and 70 °C [1].

It is also known that the cooling stage accounts for up to 80% of the injection molding process time, making it a significant factor in energy consumption [14]. Cooling energy is

required to reduce the polymer temperature inside the mold, from the melting temperature to the ejection temperature, so that the part can be removed from the mold without deformation. Recommended melting and ejection temperatures depend on the polymeric material; in turn, these temperatures depend primarily on the mass of polymer injected into the mold. To simulate the part's solidification process, a relationship exists that allows for the determination of the theoretical minimum cooling time for an injection-molded part, based on the one-dimensional heat conduction equation; Ballman and Shusman describe a model for determining the cooling time required for proper solidification as follows [15,16]:

$$t_c \text{ (sec)} = \frac{h^2}{2\pi\alpha} \ln \left(\frac{4}{\pi} \frac{T_m - T_w}{T_{eject} - T_w} \right) \quad (1)$$

where h (m) is the average wall thickness of the part, α (m²/s) is the thermal diffusivity of the material, T_m (°C) is the recommended melt temperature, T_{eject} (°C) is the recommended temperature at which the part can safely be removed from the mold, also known as the Deflection Temperature Under Load (DTUL), and T_w (°C) is the mold temperature. Busch, Field, and Rosato incorporated statistical methods to achieve Equation (2) for the estimation of the cooling time. This equation takes the part weight (W_p) and number of cavities (N_{cav}) into account [16].

$$t_c \text{ (sec)} = 1.35 \times \frac{h^2}{2\pi\alpha} \ln \left(\frac{8}{\pi^2} \frac{T_m - T_w}{T_{eject} - T_w} \right) + 0.0151W_pN_{cav} + 8.87 \quad (2)$$

In the equations presented above, the thermal diffusivity is regarded as a constant and is given by Equation (3):

$$\text{Thermal Diffusivity } (\alpha) = \frac{\text{Thermal Conductivity } (k)}{\text{Density } (\rho) \times \text{Specific Heat } (C_p)} \quad (3)$$

Based on this fundamental equation, one of the parameters used to simulate transient solidification in Ansys® Workbench for HDPE specifically, the estimated temperature at which the part is ejected, is determined; this parameter is presented below:

$$T_{eject} = \frac{(T_m - T_w)}{\frac{t_c - 0.0151W_pN_{cav} - 8.87}{\frac{1.35 \times \frac{h^2}{2\pi\alpha}}{\mu^2 e}}} + T_w \quad (4)$$

For the analysis, values defined by the material's technical data sheet and by Vieira et al. [17] used these additional physical property data, which are presented below in Table 4, to complement the calculation of thermal diffusivity, the values for the magnitudes of T_m (°C) and T_w (°C) were also defined based on the DOE, for $W_p = 0.0201$ kg, $h = 0.0033$ m, and $N_{cav} = 2$.

Table 4. Additional physical properties of HDPE.

Abbreviation	Volumetric Contraction [%]	Density [g/cm ³]	Specific Heat [J/kg K]	Thermal Conductivity [k]
HDPE	1.5–3	0.952	1900	0.45–0.52

2.6. Geometry Analysis

Parts were generated based on the established Design of Experiments, and as can be seen in Figure 5, parts were produced with visible surface depression defects in various

locations, which affect the surface quality of the parts. By obtaining different parts for different injection molding process parameters, it was found that the depth of surface depression is highly sensitive to the injection process parameters.

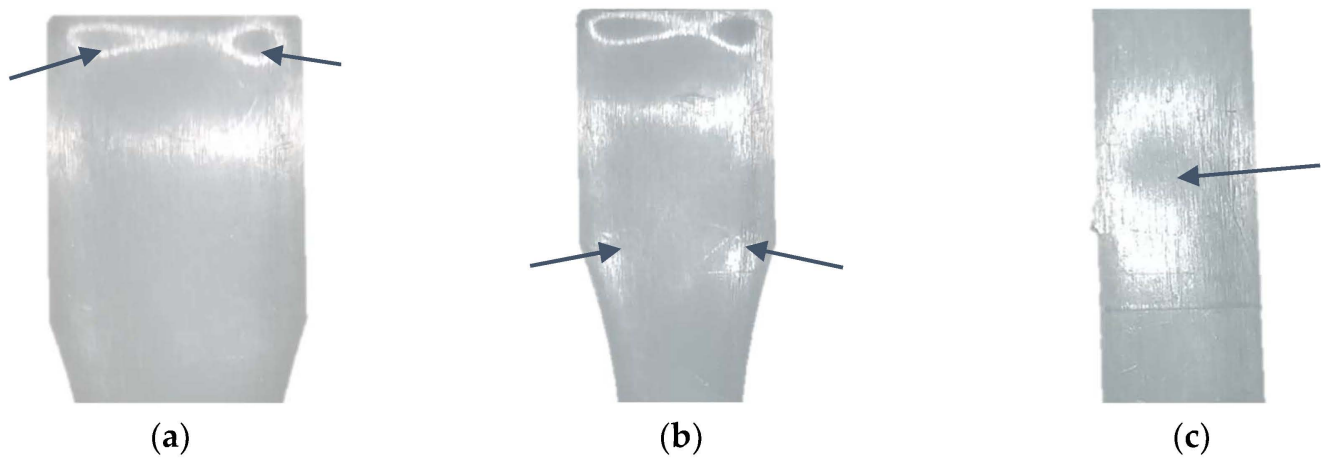


Figure 5. Injection-molded parts visually showing visible surface sink-mark defects in three different parts of the part: (a) in the upper corners, (b) side corners, and (c) in the central part of the specimen.

Once the injection molding process was completed under different conditions established in the DOE, and upon physical inspection of the resulting parts, points of greater shrinkage and deformation were identified, which are shown in Figure 6. This allowed the definition of points of interest where the results obtained by simulation with non-conventional FEA were compared with the 3D scanned parts.

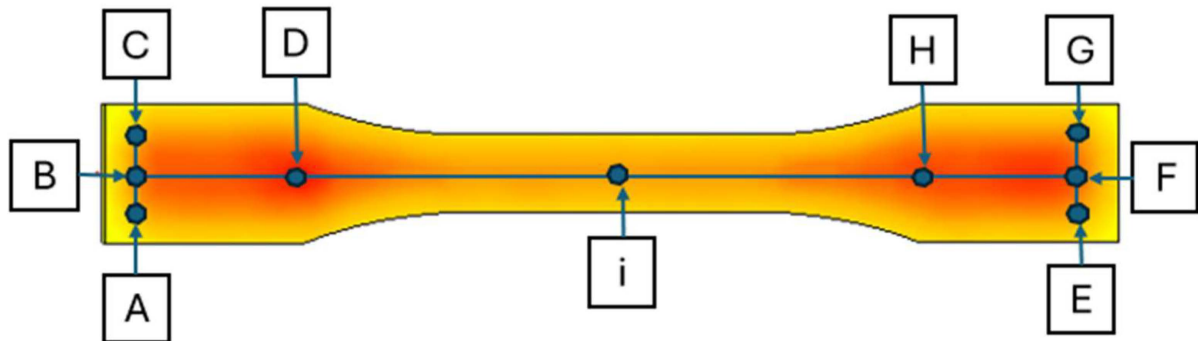


Figure 6. Locations for warpage and sink-mark measurement.

2.7. Scanning

Before performing the scanning process, a 24-h resting period was applied to ensure complete stabilization of residual stress in the parts after the injection process [18,19]. High-precision 3D scanning allows the capture of a digitized part from real-world geometry, using millions of data points. This point cloud is then used to generate a digital surface model (3D mesh). For this purpose, the ATOS triple-scan scanner (see Figure 7a), a high-resolution industrial optical 3D scanner, was used. This scanner provides fast and accurate three-dimensional measurement data to optimize automotive engineering processes and improve manufacturing workflows. To ensure the part could be scanned more accurately, the parts were painted with titanium dioxide with alcohol, since HDPE cannot be fully digitized when scanned, as seen in Figure 7b, the main features of the scanner are shown in Table 5.

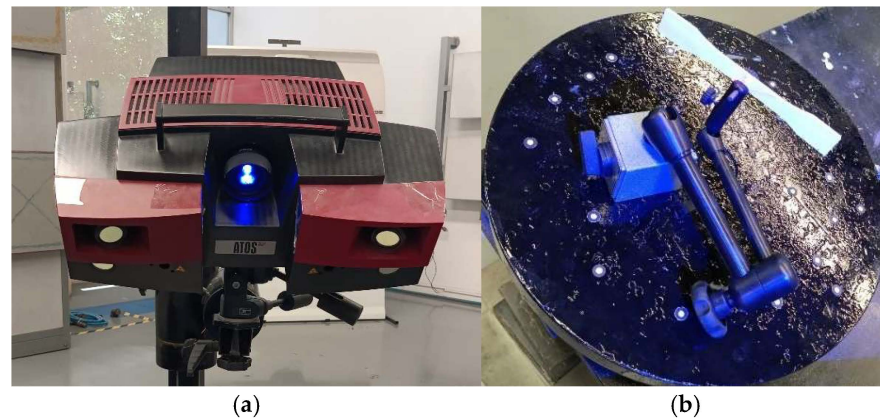


Figure 7. (a) ATOS triple scan and (b) physical scan of the injected part.

Table 5. Scanner features ATOS triple scan.

Sensor Configurations	
Camera Pixels	$2 \times 8,000,000$
Measuring Area	38 mm^2 up to 2000 mm^2
Point Spacing	0.01 mm–0.61 mm
Measured points per scan	8 million points
Operating Temperature	5–40 °C, non-condensing

Once the parts were scanned in detail, samples were taken from each of the runs generated by the DOE, and the 3D data were obtained and used to analyze their geometry and obtain deformation maps by comparing them with a part generated by CAD. This information was processed using GOM software Inspect 2019, this software is used for reverse engineering and rapid prototyping. It transforms scanned surfaces into accurate 3D numerical models using advanced geometric reconstruction algorithms. To ensure complete digitization, multiple scans were performed from various angles and then merged into a single, unified set of measurements, resulting in a detailed and comprehensive digital representation of the test specimens. After the scanning process was complete, polygonization was performed, which includes aligning all the resulting images and creating a complete numerical model. The resulting file was saved in “*.stl” format., generated with the specific software GOM Inspect, which is a procedure similar to that presented by Teodor, V.G et al. (2025) [20].

3. Results and Analysis

The methodology presented in this article has a practical orientation, unlike other similar studies that use software that mechanical design engineers generally have at their disposal to perform the analysis using CAD and finite element analysis of parts molded by the injection process. The methodology for analyzing the results obtained involves:

1. Scanning the part: A high-resolution laser or structured light scanner is used to capture the geometry of the part removed from the mold.
2. Generate a CAD part with SolidWorks Software scaling the part according to the shrinkage and shape of the CAD mold: For this, the processing guide reports that HDPE has typical shrinkage values of the order of 0.015–0.030 (1.5–3.0%). To determine the expected volumetric contraction value during the HDPE solidification phase, an estimate can be made based on the material’s melting temperature used in the injection machine

($T_m \approx 120\text{ }^{\circ}\text{C}$) and the ambient temperature ($T_a \approx 25\text{ }^{\circ}\text{C}$), resulting in $\Delta T \approx 95\text{ }^{\circ}\text{C}$; taking a typical coefficient of thermal expansion:

$$\alpha \approx 1.45 \times 10^{-4}\text{ }^{\circ}\text{C}^{-1}$$

for HDPE,

$$\varepsilon_{th} = \alpha \Delta T = (1.45 \times 10^{-4}\text{ }^{\circ}\text{C}^{-1})(95\text{ }^{\circ}\text{C}) = 0.014$$

namely

$$\varepsilon_{th} = 1.4\%$$

This aligns remarkably well with the typical linear shrinkage values observed for HDPE, which range from approximately 1% to 3%. Using this value as a starting point, the most suitable scale factor was determined by comparing the scanned parts with CAD parts adjusted via the SolidWorks scale command using various values (as shown in Figure 8); a scale factor of 2.0% was determined to be the most appropriate for the analysis in this study.

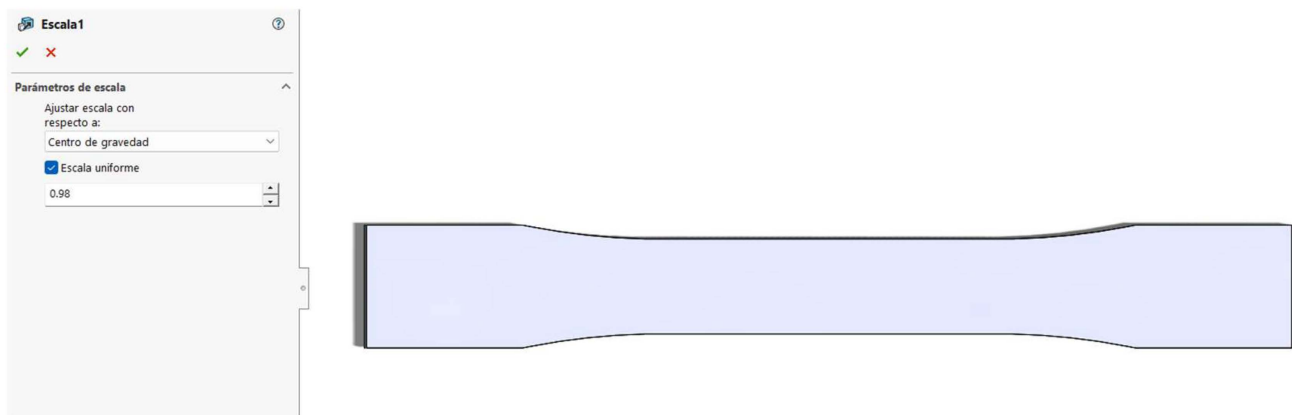


Figure 8. Geometric model of the piece, one of the samples generated by the mold.

3. CAD alignment and comparison: The scanned 3D mesh model is aligned with the original, defined, and scaled CAD model of the mold. The metrology software generates a color-coded deviation map that visually illustrates the magnitude and direction of dimensional deviations at each point on the part's surface. The series of steps is shown below:

Step 1: Processing of parts scanned in GOM Inspect 2019, as seen in Figure 9.



Figure 9. Treatment of the scanned parts.

Step 2. Splicing and processing the scanned parts for comparison with the CAD part in GOM Inspect 2019, as seen in Figure 10.



Figure 10. Splicing and treatment of the scanned parts compare with the CAD part.

Step 3: Determining the points that show the difference between the CAD part vs the scanned parts, as seen in Figure 11.

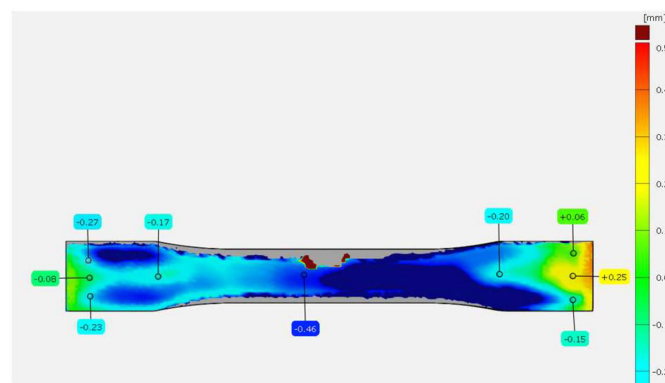


Figure 11. Determining the differences between the CAD part vs the scanned parts.

Step 4. Generate the phenomena present once the part is ejected in the Ansys® Workbench and simulate the cooling phase of the part, using equation 4 to determine the temperature at which the part is ejected (see Figure 12), for which it relies on the numerical solution, enabling predictions, sink-mark, and warping, subject to the properties of the material (PVT, thermal and mechanical properties) and the correct configuration of the software.

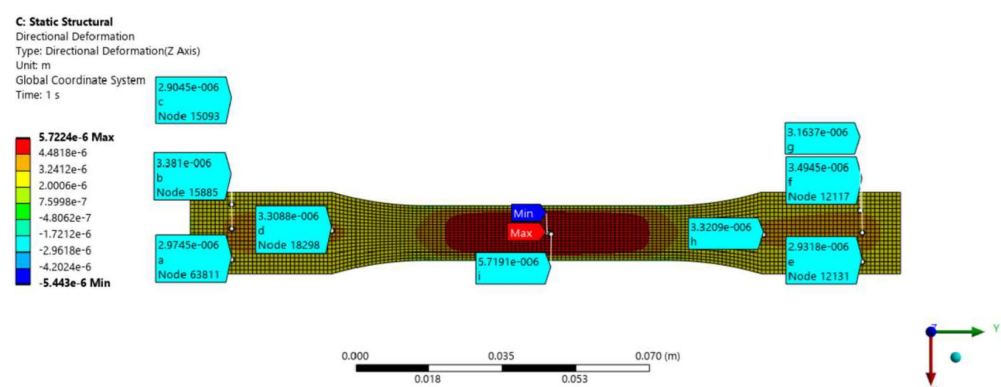


Figure 12. Sink-marks simulated by ANSYS Directional Deformation (Z axis).

Using the same solution as the analysis with Ansys® Workbench, results were also obtained from the graphs where the minimum and maximum deformation points are located, as can be seen in Figure 13, and the cross-sectional cut of these deformations is also shown, where the major directional deformations can be observed in yellow-orange and the minor ones in shades of blue.

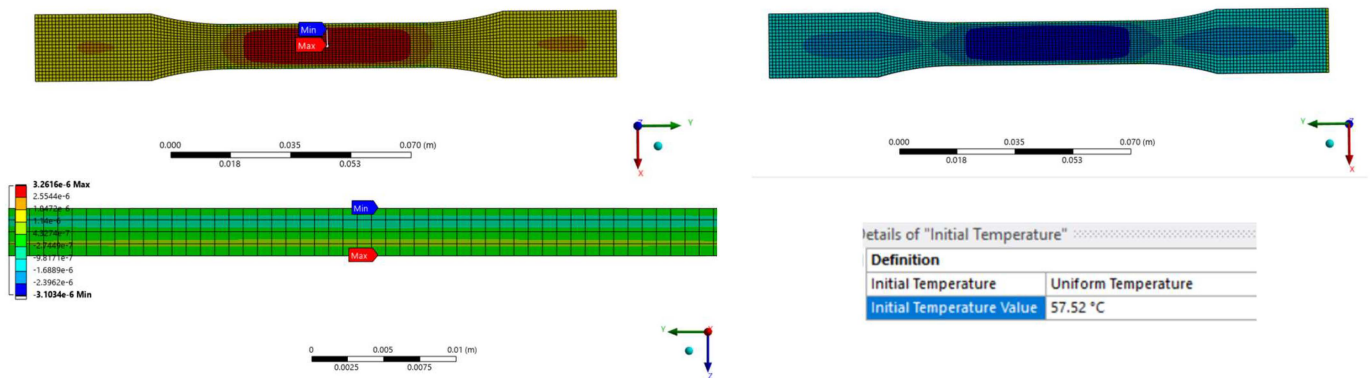


Figure 13. Front views, rear view, and cross-sectional cut view showing the deformations in the piece, simulating the cooling of the piece once it is ejected at 57.52 °C.

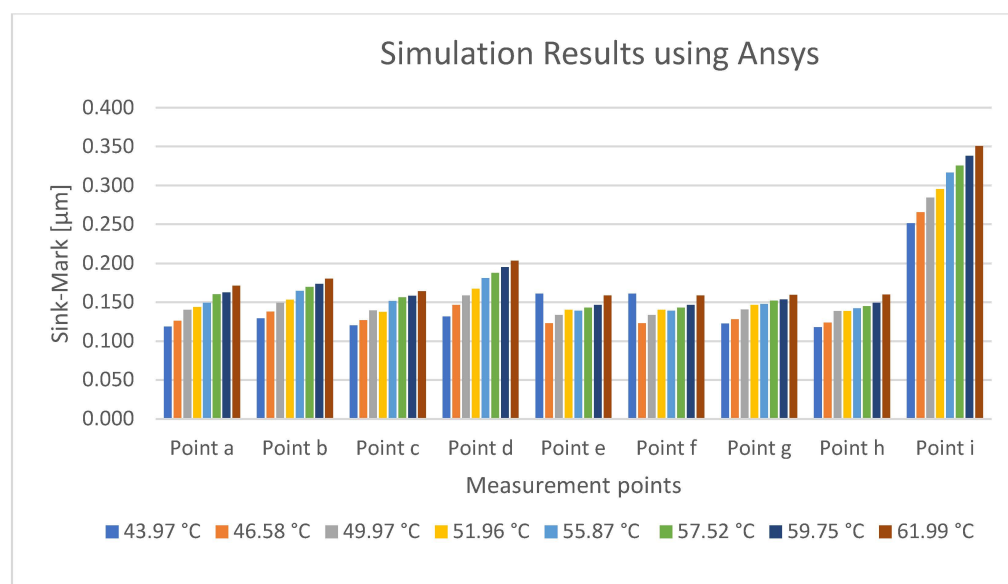
Subsequently, simulations were performed in Ansys® Workbench to calculate the part ejection temperature; using material technical data and Equation 4, calculations were carried out for all runs generated via the DOE. This yielded various temperature ranges, which were grouped into blocks with the following values: 43.97, 46.58, 49.97, 51.96, 55.87, 57.52, 59.75, and 61.99 °C. These temperature values correspond specifically to the DOE-generated runs numbered 1, 4, 10, 15, 22, 27, 29, and 30. These runs served as the basis for analyzing the values obtained via GOM Inspect 2019. The results obtained using Ansys® Workbench and GOM Inspect 2019 are presented in Figures 14a and 14b, respectively.

An analysis of Figure 14a, which shows the evolution of nine measurement points (a–i) as a function of temperature, reveals that the values tend to increase almost linearly as the temperature rises. This behavior suggests that the sink-mark responds directly to the rise in temperature, indicating a significant temperature dependence.

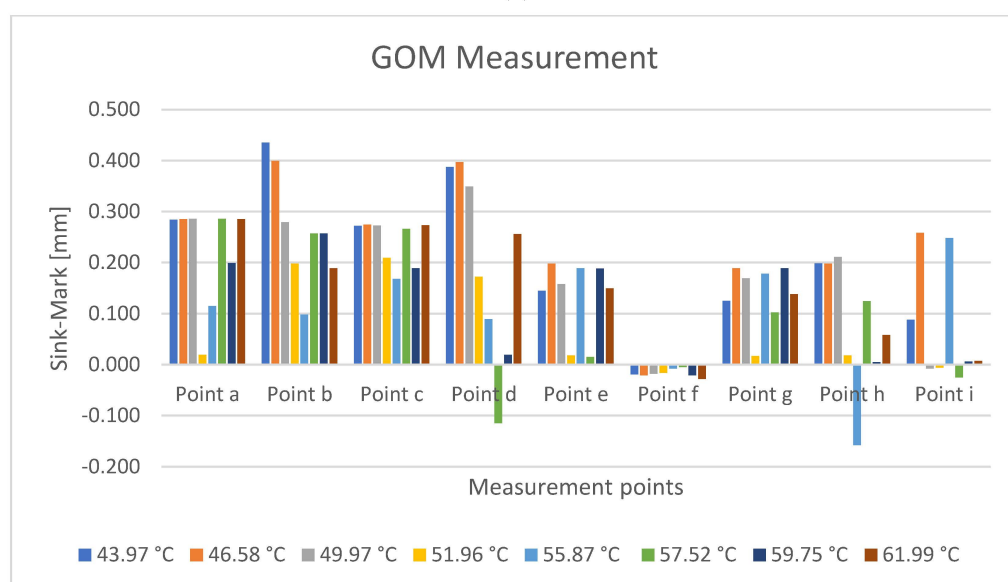
Analysis of Graph 14b, shows that the sink-mark response does not appear to follow a simple linear trend with respect to temperature. Most data points exhibit relatively high values between 44 °C and 50 °C, followed by a significant decrease around 52 °C; this suggests the occurrence of a thermal transition phenomenon or a redistribution of deformation within the part. Subsequently, some points partially recover their magnitude, while others continue to show scatter. Points A and C are the most stable and consistent throughout the temperature range, whereas points B, D, and H display the greatest sensitivity and variability. Point F exhibits a constant behavior characteristic of warpage rather than a sink-mark, indicating displacement or deformation in the direction opposite to that of a sink-mark.

Therefore, this analysis indicates that the Ansys® model reproduces a clear theoretical trend regarding temperature increase, whereas the experimental results from GOM Inspect 2019 do not fully align with that prediction. This suggests the existence of additional factors related to geometry, processing, measurement, or model configuration that may influence actual sink-mark behavior and require further detailed analysis to establish correlations between simulation results and experimental values.

Other results obtained through this methodology using Ansys® Workbench simulation show that, visually, increasing the part's ejection temperature leads to an increase in directional deformation zones during the solidification process, as indicated in Table 6.



(a)



(b)

Figure 14. Graphs of the values obtained by (a) Ansys[®] Workbench (μm) and (b) GOM Inspect 2019 (mm.).

Table 6. Graphical results obtained with Ansys[®] Workbench.

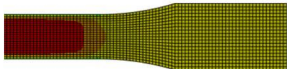
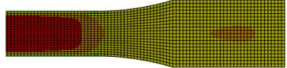
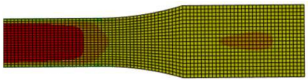
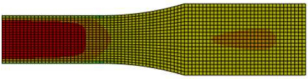
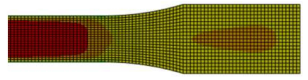
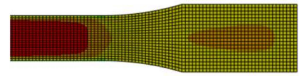
Temperature Defined in the Ansys [®] Workbench ($^{\circ}\text{C}$)	Simulation Results
43.97	
46.58	
49.97	

Table 6. *Cont.*

Temperature Defined in the Ansys® Workbench (°C)	Simulation Results
51.96	
55.87	
57.52	
59.75	
61.99	

An analysis of runs 1, 10, and 28 is also presented below in Tables 7–9, respectively, to compare the results obtained via simulation with the scanned results. These tables show some values with a “+” sign and others without a sign; the former represent measurements indicating warpages, while the latter are defined as sink-marks.

Table 7. Comparison between simulated values in ANSYS and scanned values for run 1.

Run	POINT									Sink-Mark Depth
	A	B	C	D	E	F	G	H	I	
1	0.284	0.435	0.272	0.387	0.145	+0.019	0.125	0.199	0.088	Measured (mm)
	0.119	0.1292	0.120	0.132	0.161	0.161	0.122	0.118	0.251	Simulated (μm)

Table 8. Comparison between simulated values in ANSYS and scanned values for run 10.

Run	POINT									Sink-Mark Depth
	A	B	C	D	E	F	G	H	I	
10	0.286	0.279	0.272	0.349	0.157	+0.018	0.168	0.211	+0.008	Measured (mm)
	0.140	0.149	0.139	0.159	0.133	0.133	0.140	0.139	0.284	Simulated (μm)

Table 9. Comparison between simulated values in ANSYS and scanned values for run 28.

Run	POINT									Sink-Mark Depth
	A	B	C	D	E	F	G	H	I	
28	0.199	0.257	0.189	0.019	0.188	+0.021	0.189	0.005	0.006	Measured (mm)
	0.163	0.174	0.158	0.195	0.147	0.147	0.154	0.149	0.338	Simulated (μm)

An analysis of run 1, shown in Table 7, reveals that the simulation partially reproduces the order of magnitude of the depressions but fails to accurately predict the location of the experimental maxima. The part exhibits an actual concentration of surface deformation in region B–D that the numerical model does not capture, suggesting the presence of physical phenomena not fully represented such as local thermal variations or non-uniform solidification conditions.

An analysis of run 10, shown in Table 8, reveals that the experimental behavior maintains the trend of greater subsidence in the upper-central region (A–D), whereas the simulation continues to indicate a concentration at point I. The spatial correlation between the simulation data and the experimental values is limited. However, the overall magnitude of the values is comparable, suggesting that the model partially captures the average volumetric contraction but not its local distribution.

An analysis of run 28, shown in Table 9, reveals general agreement at several points (A, B, C, and E) but retains a significant discrepancy regarding the location of the maximum sink-marks. Experimentally, the sink-marks are distributed primarily between points A, E, and G, whereas the simulation continues to concentrate the defect at point I. This indicates that the model maintains a systematic bias in the spatial prediction of the sink-mark.

Analyzing the information obtained in Tables 7–9, the following conclusions can be drawn:

An analysis of the data from Tables 7–9 leads to the following conclusions:

1. In all three experimental runs, the greatest sink-marks are generally located in regions A, B, C, and D.
2. The simulation exhibits consistent behavior, predicting maximum sink-marks at point I in every case—a result not observed experimentally.
3. The simulated results show less scatter than the measured values, indicating a more uniform model response compared to the part's actual behavior.
4. There is reasonable agreement regarding the order of magnitude at several points, particularly in run 28; however, the simulation's predictive capability for identifying critical sink-mark zones can be considered limited.
5. The observed discrepancies may be attributed to numerical model simplifications, residual stress, and geometric variations not accounted for in the simulation. These sources of error are consistent with the systematic behavior observed across the three runs.
6. The effect of the 3D scanning process; these measurements capture the final geometric response of the stabilized part, including deformations occurring after ejection and variations caused by environmental conditions.

It should be noted that the ANSYS® simulation represents only a specific stage of solidification and not necessarily the final state of the part as measured by 3D scanning; while it is useful for identifying critical zones with sink-marks, adjustments are required to predict the actual magnitude of the experimentally observed defects with greater accuracy.

The correlation coefficient was also determined using the data obtained to ascertain the strength and direction of the linear relationship between two variables. For this study, the variables were compared between casting temperature, mold temperature, pressure, and holding time, and the results obtained from the differences found between the scanned part and the CAD model. These factors are shown in Figure 15.

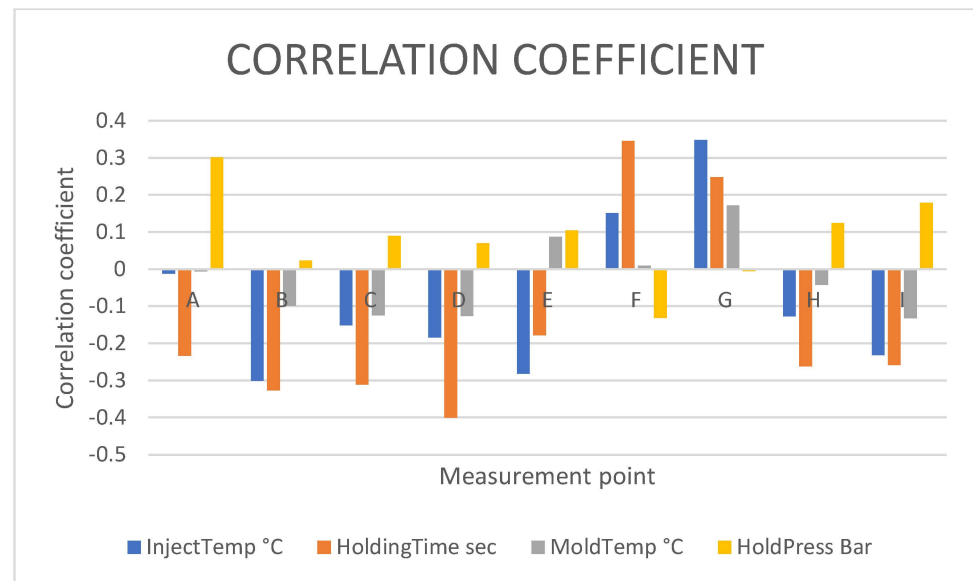


Figure 15. Correlation factors for the different points.

A correlation study measures the strength of the relationship between different variables. Below is an interpretation of how each parameter impacts this type of analysis:

(a) Holding Time is the most influential parameter, as it exhibits the largest correlation coefficients across several categories:

- Negative for A, B, C, D, E, and H.
- Positive for F and G.

This indicates that it is the process parameter most likely to affect the analyzed results.

(b) Inject Temp ranks second in importance, showing:

- Negative correlations for B, C, D, E, and H.
- Positive correlations for F and G.

Its effect varies depending on the characteristic being evaluated.

(c) Mold Temp has minimal influence, as most of its coefficients are close to zero.

Mold temperature appears to be the least critical parameter within the evaluated ranges.

(d) Hold Press has selective effects:

- Positive for A, E, and H.
- Negative for F.
- Negligible for G.

It appears to influence only specific responses.

An analysis of these results suggests that Holding Time is the most important process factor, followed by Inject Temp. Mold Temp makes a limited contribution, and Hold Press affects only certain responses. Another conclusion is that most responses center on relatively stable values, suggesting the process is not significantly unstable; the correlations are predominantly negative for points A, B, C, D, E, H, and I, whereas they are positive for points F and G.

Based on the graph, quality improvement efforts could focus initially on the variables showing the strongest correlation with process responses: Holding Time and Inject Temp.

4. Conclusions

This study presents a practical analysis of various methods for examining sink-marks and warpage defects in HDPE molded parts after ejection specifically the use of Ansys® Workbench software, and the results that can be obtained with such tools. This also shows the steps to follow for using the GOM Inspect 2019 software to compare the scanned parts generated by the injection process, which were generated by different parameters, creating parts with intentional defects, and comparing them with the CAD part that should theoretically be obtained. The conclusions can be drawn as follows:

1. The clearest signal from the DOE is that Holding Time is the most influential factor overall.
2. Increasing the Holding Time tends to make A, B, C, D, E, H, and I more negative, resulting in more volumetric contractions in the part compared to the CAD model.
3. Increasing the Holding Time tends to make F and G more positive, resulting in slight valleys in the part compared to the CAD model.
4. Specifically, the main factors that influence failures in the injection molding process, such as deformation and shrinkage, for comparative analysis are:

Mold temperature: Influences the cooling rate and the distribution of internal stresses.

Holding pressure: Affects the material's ability to compensate for warpage.

Polymer type: HDPE can exhibit varying degrees of warpage and thermal behavior.

Part geometric design: Areas with abrupt thickness changes are more prone to sink-mark.

5. Visually, using the ANSYS® Workbench, it can be predicted that once the part is ejected, as the part's temperature rises above the ambient temperature, directional deformations appear and increase in the part, in areas where deformations also appear in the scanned parts.

They are found on finished parts as depressions on the surface, formed in the final stage of the plastic injection process during the cooling process. This is because the thicker sections of the part cool at a slower rate than the other sections of the molded part. Then the piece on the outside cools and solidifies and the material inside begins to cool and its volumetric contraction pulls the surface inward, causing a depression in the surface of the piece.

The combination of thermo-structural analysis of the solidification phase and 3D scanning represents the evolution of deformation analysis in injection molding. Simulations using unconventional software offer a different perspective on the fluid-thermal-structural interactions within the mold.

Simultaneously, 3D scanning provides high-fidelity experimental validation, which is crucial for designing products with stringent accuracy requirements. This combination of methodologies not only helps identify the causes of deformation (cooling differential, warpage, or sink-mark) but also enables the optimization of injection molding process conditions, the adjustment of injection parameters such as holding pressure, mold temperature, material melting temperature, cooling time, etc., and the rational design of the mold.

Author Contributions: Conceptualization, I.R.-L.; Methodology, V.Z.-J., H.P.-M., J.N.D., J.F.R.A. and M.C.L.-M.; Software, V.Z.-J. and I.R.-L.; Validation, I.R.-L. and J.F.R.A.; Investigation, V.Z.-J., H.P.-M., J.N.D., I.R.-L. and J.F.R.A.; Visualization, V.Z.-J. and H.P.-M. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding authors.

Acknowledgments: Industrial Engineering Laboratory at the Technological Institute of San Juan del Río, Querétaro, Mexico. PEMSA (Pintura Estampado y Montaje S.A.P.I. de C.V) en Celaya, Guanajuato, México.

Conflicts of Interest: Author Ismael Ruiz-López was employed by the company Pintura Estampado y Montaje S.A.P.I. de C.V. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

References

1. Chakravorty, S. PVT testing of polymers under industrial processing conditions. *Polym. Test.* **2002**, *21*, 313–317. [CrossRef]
2. Kościuszko, A.; Rojewski, M.; Nowinka, B.; Patalas, F. Post-Molding Shrinkage, Structure and Properties of Cellular Injection-Molded Polypropylene. *Materials* **2022**, *15*, 7079. [CrossRef]
3. Plastics Europe. Plastics—The Fast Facts 2024—Plastics Europe ES. Available online: <https://plasticseurope.org/es/knowledge-hub/plastics-the-fast-facts-2024/> (accessed on 10 July 2025).
4. Huang, M.-C.; Tai, C.-C. The effective factors in the warpage problem of an injection-molded part with a thin shell feature. *J. Mater. Process. Technol.* **2001**, *110*, 1–9. [CrossRef]
5. Knoll, J.; Heim, H.-P. Analysis of the Similarity between Injection Molding Simulation and Experiment. *Polymers* **2024**, *16*, 1265. [CrossRef]
6. Fan, Z.; Lin, B.; Costa, F.; Jin, X.S.; Zheng, R.; Kennedy, P. Three-dimensional warpage simulation for injection molding. *Annu. Tech. Conf. ANTEC Conf. Proc.* **2004**, *1*, 491–495.
7. Lin, W.-C.; Fan, F.-Y.; Huang, C.-F.; Shen, Y.-K.; Wang, H. Analysis of the Warpage Phenomenon of Micro-Sized Parts with Precision Injection Molding by Experiment, Numerical Simulation, and Grey Theory. *Polymers* **2022**, *14*, 1845. [CrossRef]
8. Godec, D.; Pand̄a, F.; Tujmer, M.; Monkova, K. Molded Part Warpage Optimization Using Inverse Contouring Method. *Polymers* **2025**, *17*, 2278. [CrossRef]
9. Hakimian, E.; Sulong, A.B. Analysis of warpage and shrinkage properties of injection-molded micro gears polymer composites using numerical simulations assisted by the Taguchi method. *Mater. Des.* **2012**, *42*, 62–71. [CrossRef]
10. Tillmann, S.; Behr, M.; Elgeti, S. Using Bayesian optimization for warpage compensation in injection molding. *Mater. Werkst.* **2024**, *55*, 13–20. [CrossRef]
11. Huang, Y.; Huang, C.; Peng, B.; Chang, C.; Cheng, H.; Lin, Y.; Shen, Y.; Wang, H. Experimental and Numerical Study Determining the Warpage Phenomenon of Thin-Wall Injection Molding. *Adv. Polym. Technol.* **2020**, *2020*, 1–13. [CrossRef]
12. Kurtaran, H.; Erzurumlu, T. Efficient warpage optimization of thin shell plastic parts using response surface methodology and genetic algorithm. *Int. J. Adv. Manuf. Technol.* **2005**, *27*, 468–472. [CrossRef]
13. Páramo, J.J.; Reveles Arredondo, J.F.; Mora, H.P.; Gómez, E.A. Análisis de un proceso de inyección de plástico por interacción fluido estructural y cambio de estado. *Acta Univ.* **2019**, *29*, 1–18. [CrossRef]
14. Rashid, O.; Low, K.; Pittman, J. Mold cooling in thermoplastics injection molding: Effectiveness and energy efficiency. *J. Clean. Prod.* **2020**, *264*, 121375. [CrossRef]
15. Moussa, K.; Takash, A.A.; Hassan, A.; Faraj, J.; Khaled, M.; Salameh, T.; Hamid, A.; Hussein, M. Optimizing rib design in injection-molded plastic containers for enhanced strength and reduced molding energy consumption. *Next Mater.* **2025**, *10*, 101469. [CrossRef]
16. Fitzgerald, A.; McDonald, P.; Devine, D.; Fuenmayor, E. Transfer and Optimisation of Injection Moulding Manufacture of Medical Devices Using Scientific Moulding Principles. *J. Manuf. Mater. Process.* **2021**, *5*, 113. [CrossRef]
17. Vieira, A.; Campilho, R.; Silva, F.; Faria, N.; Ferreira, L. Design of a thermoplastic micro over injection machine for the automotive component industry. *Procedia Manuf.* **2021**, *55*, 56–63. [CrossRef]
18. Jiménez-Armendáriz, J.; Guevara-Morales, A.; Figueroa-López, U.; Alfaro-Ponce, M.; Martínez-Trinidad, J.; Jimenez-Martinez, M. Optimization of Injection Molding Parameters for Warpage Reduction on Polypropylene Plates. *J. Manuf. Mater. Process.* **2025**, *9*, 393. [CrossRef]
19. Sun, X.; Zeng, D.; Tibbenham, P.; Su, X.; Kang, H. A new characterizing method for warpage measurement of injection-molded thermoplastics. *Polym. Test.* **2019**, *76*, 320–325. [CrossRef]
20. Teodor, V.G.; Baroiu, N.; Păunoiu, V. Profiling the Outer Rotor of a Conical Helical Compressor via Kinematic Simulation and Experimental Validation. *Modelling* **2025**, *6*, 143. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.