EUROPEAN MECHANICAL SCIENCE

Research Article

e-ISSN: 2587-1110

Analysis of the pressure resistance of mini valve used in drip irrigation with finite elements

Yavuz Turan[1], **Murat Kuru**[1], **Beyza Gizem Duman**[1], **Hülya Alaca**[1], **İbrahim Keleş**[2*]

[1] Poelsan Plastic Industry and Trade Inc., 55139, Tekkeköy/Samsun, Türkiye

[2] Samsun University, Faculty of Engineering and Natural Sciences, Department of Mechanical Engineering, 55420, Samsun, Türkiye

Abstract: Different types of valves are used in irrigation systems to transport water efficiently. One of these valves is mini valves, which attract attention with their ease of use and functionality. Mini valves are elements that can be quickly mounted and dismounted on the irrigation line and are generally preferred for flow control and on/off operations. In the design of valves, it is of great importance to analyze the pressure losses occurring in the irrigation system. In this study, the strength of a manually controlled, two-way mini plastic valve, which is frequently used in irrigation systems, is numerically investigated. This mini valve is an element used to control the flow direction of water in a certain direction and is especially used in drip irrigation systems. These valves, which are preferred to minimize pressure losses and manually manage the flow of the line, play a critical role in terms of system efficiency. Valves must be resistant to certain pressure values for long life and safe operation. In this study, a manually controlled bi-directional mini plastic valve with a worm mouth connection port was designed and its resistance was evaluated by numerical analysis. Tests were performed by applying 2, 4 and 6 bar pressure to the valve respectively. According to the results of the analysis, the mini valve can operate safely up to 2 bar pressure, but it is seen that the stress levels increase as the pressure increases. At 6 bar pressure, it was found that the valve approached the material strength limit. These findings provide valuable information to determine the operating limits required for the safe use of mini plastic valves in irrigation systems.

Keywords: Mini valve, Finite element analysis, Compressive strength, Stress analysis

1. Introduction

Valves are critical components used in pipelines to regulate, direct or stop the flow of fluids. Although different types of valves are used in industrial and agricultural applications, low-pressure, lightweight and durable mini plastic valves are generally preferred in drip irrigation systems. These valves provide controlled distribution of water in irrigation lines and offer advantages in terms of energy efficiency and cost. However, their structural strength is of great importance due to the pressure and mechanical loads they are exposed to during operation. The performance of mini plastic valves is affected by several hydraulic, mechanical and environmental factors. From a hydraulic point of view, pressure loss directly affects energy efficiency due to the flow resistance inside the valve, while flow and flow characteristics vary depending on the valve design. High turbulence can lead to energy loss and wear, while cavitation in low pressure zones can threaten the structural integrity of the valve. Among mechanical factors, the strength and sealing performance of the plastic material used are critical. In long-term use, there is a risk of material fatigue and deformation under high pressure. Environmental factors are also decisive for the durability of valves; temperature changes can cause expansion and contraction of plastic materials, while fertilizers and chemicals used in agricultural irrigation can affect chemical resistance. In addition, structural weakening of plastic materials exposed to UV radiation can occur over time. In terms of installation and use, the way the valve is connected to the pipe and the sensitivity of the on-off mechanism are important for the overall efficiency of the system. Therefore, the determination of flow parameters such as pressure loss, flow coefficient, resistance coefficient and cavitation index, which are critical in the design process of valves, is necessary to understand their flow behavior.

*Corresponding author:
Email: ibrahim.keles@samsun.edu.tr

Cite this article as:

Turan, Y., *et.al.* (2025). Analysis of the pressure resistance of mini valve used in drip irrigation with finite elements. *European Mechanical Science*, 9(1): 87-95. https://doi.org/10.26701/ems.1646687

History dates:

Received: 25.02.2025, ***Revision Request***: 17.03.2025, ***Last Revision Received***: 18.03.2025, ***Accepted***: 18.03.2025

Rajkumar [1] identified valve design problems encountered in industry and showed that by using finite element analysis (FEA) and computational fluid dynamics (CFD) methods, the flange weight was reduced by 50% while stress and deformation intensities remained within acceptable limits. Chem M. J. et al. [2] experimentally investigated inlet and outlet flow patterns, cavitation effects and valve performance coefficients in ball valves using Particle Tracking Visualization Method (PTVM) and showed that the proposed method provides rapid effect evaluation. Jadhav S. S. [3] determined the optimum thickness specifications for safe operation of M-Type gate valve under high pressure and showed that the stress values obtained by classical mechanical theory and FEA agree and these results can be used for further improvement of the valve. Considering that cavitation can lead to noise and vibrations due to localized under pressure, this phenomenon needs to be well understood during valve operation. Furthermore, since valves are the limiting components in any flow system, their design and performance analysis are critical [4]. Mokhtarzadeh-Dehghan et al. [5] analyzed the laminar flow in a differential angle hydraulic pressure relief valve for variable compression ratio pistons using the Finite Element Method and compared the velocity and pressure distributions, the buoyancy forces on the piston and the effect of recirculation zones on the pressure distribution with experimental and analytical data. Song X. et al. [6] proposed a meta-model for optimization of butterfly valve and used kriging model to obtain the best disc weight by FEA and CFD analysis under the constraints of pressure loss coefficient and disc safety and reduced the weight of the valve disc by 7.05% through the optimization process. Kumar S. S. et al. [7] modeled the body and seat ring of a gate valve with SolidWorks and investigated factors such as stress analysis, temperature distribution, total deformation and heat flux with ANSYS analysis tool and evaluated the effects of directed heat flux on temperature distribution and welding defects such as cracking, deformation and undercutting. Prakash et al. [8] CFD analysis supports the product development process by investigating ball and gate valves with parameters such as pressure, density, viscosity and temperature, combining pressure distribution and deformations in the valve system with finite element analysis to optimize the integration of material and product design. Song X. et al. [9] optimized the valve disc by reducing the weight of the disc by 7.05% through FEA of the mechanical properties of the disc, determination of the pressure loss coefficient by fluid analysis and interpolation of the data with the Kriging model, making the optimization process more efficient. Raut L. B. et al. [10] performed FEA analysis for weight reduction of plug valve body, used strain gauge technique for stress measurement and developed optimized models with design parameters and validated the results obtained with maximum deviations of 9.75% and minimum deviations of 6.23%. Lee et al. [11] improved the flow performance of the check valve by optimizing the flow coefficient for the pan check valve by CFD analysis, determining the length of the support beam by pressure drop analysis and estimating the flow head loss at different flow rates, observing the pressure and velocity distribution using CFD method, and determining the optimum support beam length as 32 mm for minimum pressure drop and maximum flow coefficient. Jun-Oh K. et al. [12] emphasized the importance of topology optimization in the butterfly valve shape design process and explained how to optimize the topology of the double eccentric butterfly valve disc and how they used this optimization to define the disc shape and pressure drop was observed in the comparison between the original design and the best design.

While numerous studies have focused on the structural optimization, weight reduction, and flow behavior of industrial and metal-based valves, research specifically targeting the structural integrity and pressure resistance of mini plastic valves used in agricultural irrigation systems remains limited. Unlike previous works, this study provides a comprehensive numerical investigation of a mini plastic valve, assessing its mechanical strength under different pressure conditions using FEA. This research contributes to the field by offering new insights into the operational limits and failure mechanisms of plastic valves commonly used in drip irrigation systems.

Although various valve designs have been analyzed in previous studies, there is limited research on the structural analysis of mini plastic valves specifically used in agricultural irrigation systems. This study provides a detailed FEA of a mini plastic valve, evaluating its stress distribution, deformation, and safety limits under different pressure conditions. Unlike prior works, this research focuses on the mechanical strength and failure behavior of mini plastic valves, addressing a crucial gap in the literature. Unlike conventional mini valves, this study presents a novel valve design with several innovative features to enhance efficiency and durability in agricultural irrigation systems. The primary innovation includes the integration of a worm mouth connection port for improved sealing, optimized internal flow channels to reduce pressure losses, and a reinforced snap-fit mechanism in the handle connection for enhanced mechanical stability. Additionally, the material selection—Polyoxymethylene (POM) for strength and chemical resistance, and Polypropylene (PP) for flexibility and UV durability—ensures long-term reliability under outdoor conditions. These features collectively provide a more robust, efficient, and user-friendly solution compared to existing designs. The findings of this study not only provide a better understanding of the mechanical limitations of mini plastic valves but also offer practical insights for manufacturers to improve design durability and optimize operational performance.

In this study, the FEA analysis of a mini-valve with a water passage diameter of 10.4 mm, which has a claw structure on the on-off lever and opens and closes by rotating on these claws, was analyzed with SolidWorks simulator. The valve body material is POM an engineering plastic also known as polyacetal or acetal, and the valve opening-closing handle material is polypropylene. Within the scope of this investigation, three different compressive strengths of the valve under 2, 4 and 6 bar pressure were examined, and graphs were taken to evaluate the changes.

2. Materials and Methods

POM, also known as acetal, was selected for the valve body due to its high mechanical strength, dimensional stability, and excellent resistance to moisture and chemicals. POM is widely used in fluid control applications because of its low friction coefficient and superior fatigue resistance. PP was chosen for the handle due to its lightweight nature, high impact resistance, and cost-effectiveness. PP also provides good resistance to environmental factors such as UV exposure, which is critical for outdoor agricultural applications. POM exhibits semi-crystalline behavior with a balance of strength and toughness. It is considered a ductile material under normal operating conditions but can exhibit brittle fracture when exposed to high strain rates or low temperatures. On the other hand, PP is also a ductile polymer, known for its ability to withstand impact without significant deformation. Given their ductile nature, von Mises failure criteria were used to evaluate stress distribution and structural integrity.

In this study, the mini valve shown in ▶**Figure 1**, which has a length of 147 mm and a height of 42 mm, with a water passage diameter of 10.4 mm, has a clawed structure with an opening and closing handle and opens and closes by rotating on these claws, was examined. Three different pressure values were applied to the surface pressure of the mini valve to be analyzed in the working environment using the SolidWorks program [13]. The valve body is made of POM acetal, and the opening-closing handle is made of PP, and the mechanical properties of these plastics are given in ▶**Table 1**. The newly designed mini valve introduces a worm mouth connection system, which enhances the sealing efficiency and ensures a more secure fit within the irrigation pipeline. The internal flow passages were redesigned using finite element-based topology optimization to minimize turbulence and pressure drop, thus improving overall efficiency. The reinforced snap-fit mechanism in the handle connection is designed to prevent mechanical failure under repeated use, a common issue in conventional designs. These modifications provide improved structural integrity and longer service life, making the valve highly suitable for real-world agricultural applications.

Table 1. Mechanical properties of the materials that make up the mini valve for analysis

Material Feature	POM Asetal	PP
Modulus of Elasticity (N/mm^2)	2600	896
Poisson Ratio	0.3859	0.4103
Tear Modulus (N/mm^2)	932.8	315.8
Bulk Density (kg/m^3)	1390	890
Tensile Strength (N/mm^2)	71.5	27.6
Thermal Conductivity W/(m-K)	0.221	0.147
Specific Heat J/(kg-K)	1378	1881

2.1. Finite Element Modeling

In this study, a linear elastic material model was assumed for both POM and PP materials. Given that the valve operates under relatively low-pressure conditions (maximum 6 bar), nonlinearity due to plastic deformation or large deformations was not considered in the simulations. The von Mises stress criterion was used to evaluate the structural performance, assuming that both materials behave elastically within the analyzed pressure range.

The simulations were conducted using a linear static FEA in SolidWorks. The governing equation for static equilibrium in elasticity is given by:

$$\nabla . \sigma + F = 0 \tag{1}$$

where σ represents the stress tensor and F is the applied force per unit volume. Since the study assumes small

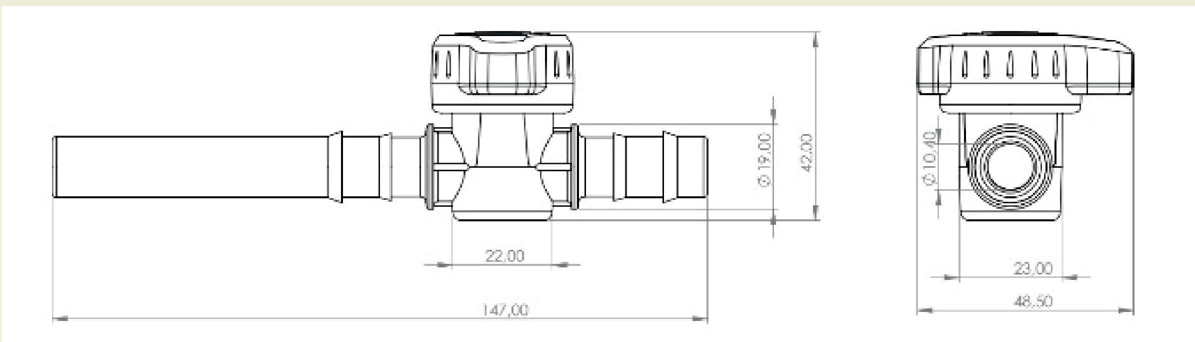


Figure 1. Mini valve used in the study

 https://doi.org/10.26701/ems.1646687

deformations and elastic behavior, the constitutive equation follows Hooke's Law:

$$\sigma = D.\varepsilon \tag{2}$$

where D is the elasticity matrix and ε is the strain tensor. The von Mises yield criterion was employed to determine stress distribution and failure risk, which is defined as:

$$\sigma_V = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \tag{3}$$

2.2. Consideration of Hoop Stress in the Mini Valve Design

Hoop stress (σ_h) is a critical factor in cylindrical pressure vessels and pipeline structures, where internal pressure generates circumferential stresses along the walls. The general equation for hoop stress in a thin-walled pressure vessel is given by:

$$\sigma_H = \frac{P.r}{t} \tag{4}$$

where P is the internal pressure, r is the inner radius, and t is the wall thickness.

In conventional pipe systems, hoop stress dominates the stress distribution, but the geometry of the mini valve differs from a simple cylindrical shell. The mini valve contains complex features such as threaded inlet/outlet connections, a central flow regulation chamber, and varying wall thicknesses, making direct hoop stress calculations less straightforward.

To account for circumferential stress effects, the FEA model includes:

- Fine meshing in the inlet/outlet regions, where pressure-induced stresses resemble hoop stress in cylindrical structures.
- Stress distribution analysis along the valve body and internal walls, ensuring that regions experiencing circumferential loading are adequately captured.
- Von Mises stress evaluation, which inherently incorporates principal stresses, including hoop stress components.
- While hoop stress was not explicitly isolated as a separate parameter, its effects are embedded within the FEA results.

The geometry of the mini valve is given in ▶Figure 2. In the simulation, the mini valve is positioned to be fully open. The boundary condition was determined by connecting the elements in contact with each other at the contact surfaces.

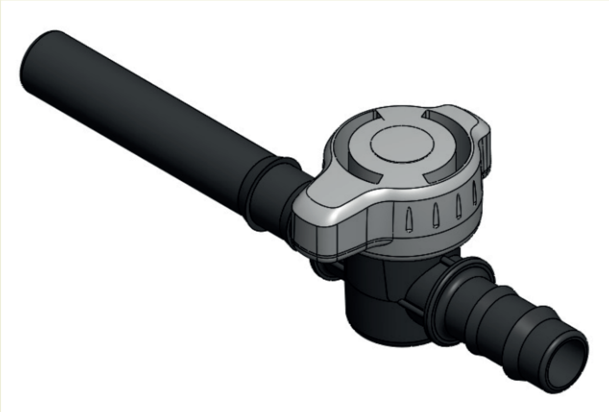


Figure 2. Geometric shape of the mini valve.

▶Figure 3 shows the boundary conditions applied to the mini valve. The inlet and outlet parts of the valve have threaded structures that provide the connection. Since these parts will be connected to female fittings in practice, a fixed boundary condition is defined for the inlet and outlet regions (▶Figure 3 (A)). In ▶Figure 3 (B), a pressure boundary condition of 2,4,6 bar is defined for the regions shown in red color.

2.3. Justification of Boundary Conditions and Pressure Levels

The applied pressure values (2, 4, and 6 bar) were selected based on realistic operating conditions in agricultural drip irrigation systems. According to industry standards and manufacturer specifications, mini plastic valves used in such systems typically operate within a pressure range of 1.5 to 5 bar, depending on the pipeline configuration and system design.

2 bar pressure represents a common working pressure in standard drip irrigation applications, ensuring optimal water distribution. 4 bar pressure was chosen as an intermediate scenario to analyze potential mechanical responses under increased load. 6 bar pressure serves as an upper limit test, allowing us to evaluate the structural integrity of the valve under extreme but possible conditions, particularly during unexpected pressure surges or water hammer effects.

2.4. Boundary Condition Application in FEA

Fixed constraints were applied at the inlet and outlet to represent real-world installation, where the valve is secured within the pipeline. A uniform internal pressure was applied to the internal surfaces of the valve to simulate hydraulic loading conditions. The meshing was refined in stress-critical regions (snap-fit connection and flow chamber walls) to ensure accurate stress predictions under applied boundary conditions. These boundary conditions and pressure selections ensure that the simulation results closely represent practical operational conditions in agricultural irrigation systems.

(A) (B)

Figure 3. Boundary conditions of the analyzed mini valve A) fixing surfaces and B) pressure application surfaces

For the numerical solution of the model shown in ▶**Figure 4**, shell elements are defined via the Shell Definition feature. Specifically: Thin Shell (equivalent to SHELL181 in ANSYS) was used for thin-walled sections to account for bending and membrane effects. Also, Thick Shell (equivalent to SHELL281 in ANSYS) was applied in areas requiring improved curvature accuracy. These shell elements were subjected to curvature-based high-quality meshing due to the shape of the geometry using triangular cells with 147249 nodes and 87444 elements to ensure correct stress distribution while maintaining computational efficiency.

2.5. Mesh Convergence Analysis and Validation

To ensure that the obtained results are mesh-independent, a mesh convergence study was conducted by systematically refining the mesh and monitoring the change in maximum von Mises stress values. The analysis was performed using three different mesh densities:

Coarse Mesh: ~75,000 elements

Medium Mesh: ~87,444 elements (selected for final analysis)

Fine Mesh: ~120,000 elements

The results showed that beyond 87,000 elements, the variation in maximum von Mises stress was less than 2%, indicating that the solution had converged. Thus, the medium-density mesh was selected to balance computational efficiency and accuracy.

2.6. FEA Validation Approach

Since experimental validation was not within the scope of this study, a basic FEA validation approach was employed:

- The maximum von Mises stress values from the simulation were compared with analytical hoop stress estimates for cylindrical sections of the valve.
- The stress distribution along the snap-fit connection and valve body was analyzed to ensure expected stress patterns were observed.
- The average stress values from mesh refinement studies were compared to verify mesh-independent results.

These validation steps confirm that the simulation results are reliable and mesh-independent, strengthening the accuracy of the presented findings.

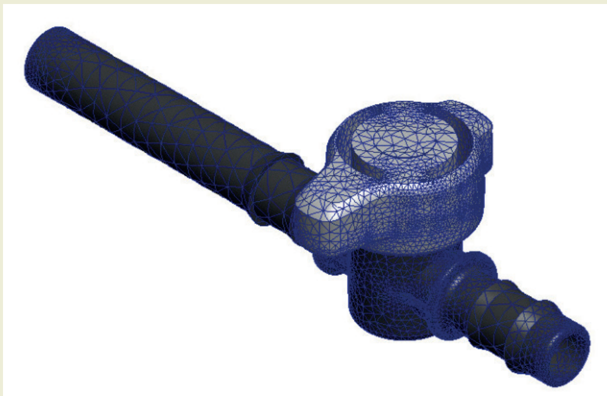


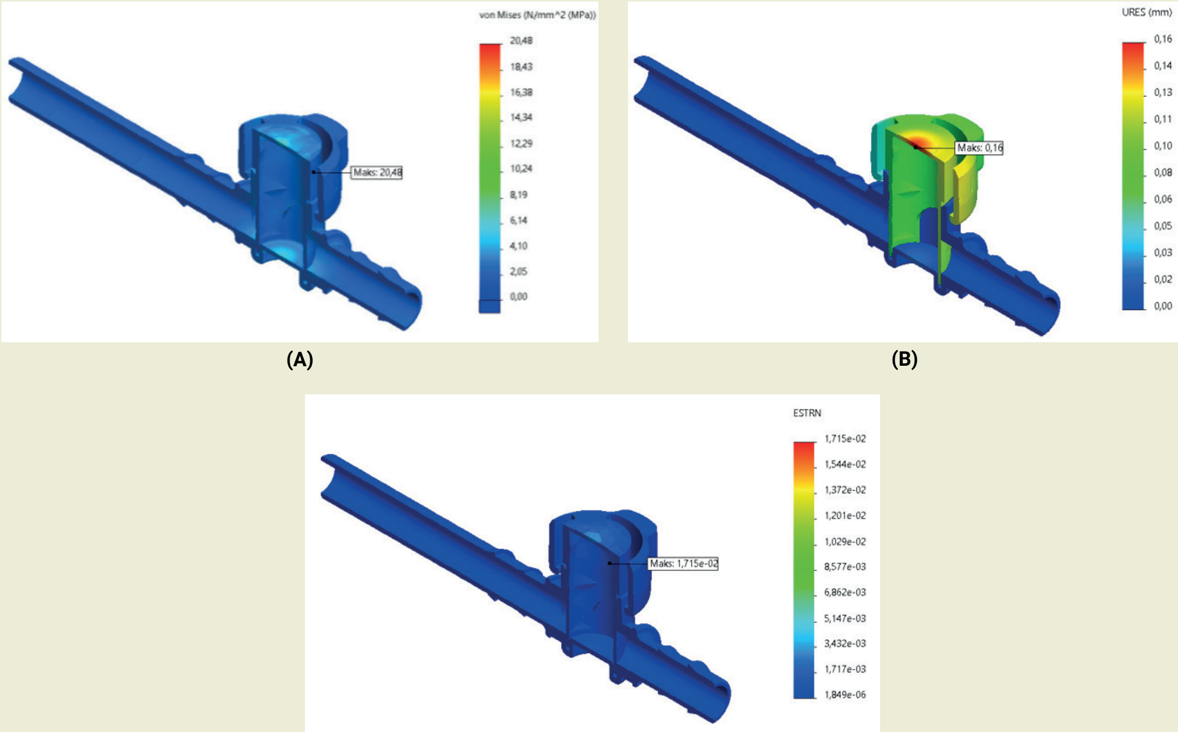
Figure 4. FEA Model meshing of the mini valve analyzed

3. Results and Discussions

To determine the compressive strength of the plastic mini valve, stresses, displacements and strains at 2, 4, 6 bar pressure values were tried to be determined. In line with this scope, Von Mises stresses, displacements and strains are shown in ▶**Figure 5** by applying 2 bar pressure to the relevant surface of the mini valve.

Figure 5 (A) shows that the maximum stress in the mini valve occurs at the edges of the snap fitting between the valve body and the valve opening and closing handle. This polypropylene material is lower than the maximum flexural yield strength of 44.1 MPa. With a maximum stress value of 20 MPa obtained at 2 bar pressure, the valve has a maximum displacement of 0.16 mm as shown in ▶**Figure 5** (B). Again, under this pressure, the maximum strain occurs in the plug-in fastener with a value of 1.75x10^{-2} (▶**Figure 5** (C)).

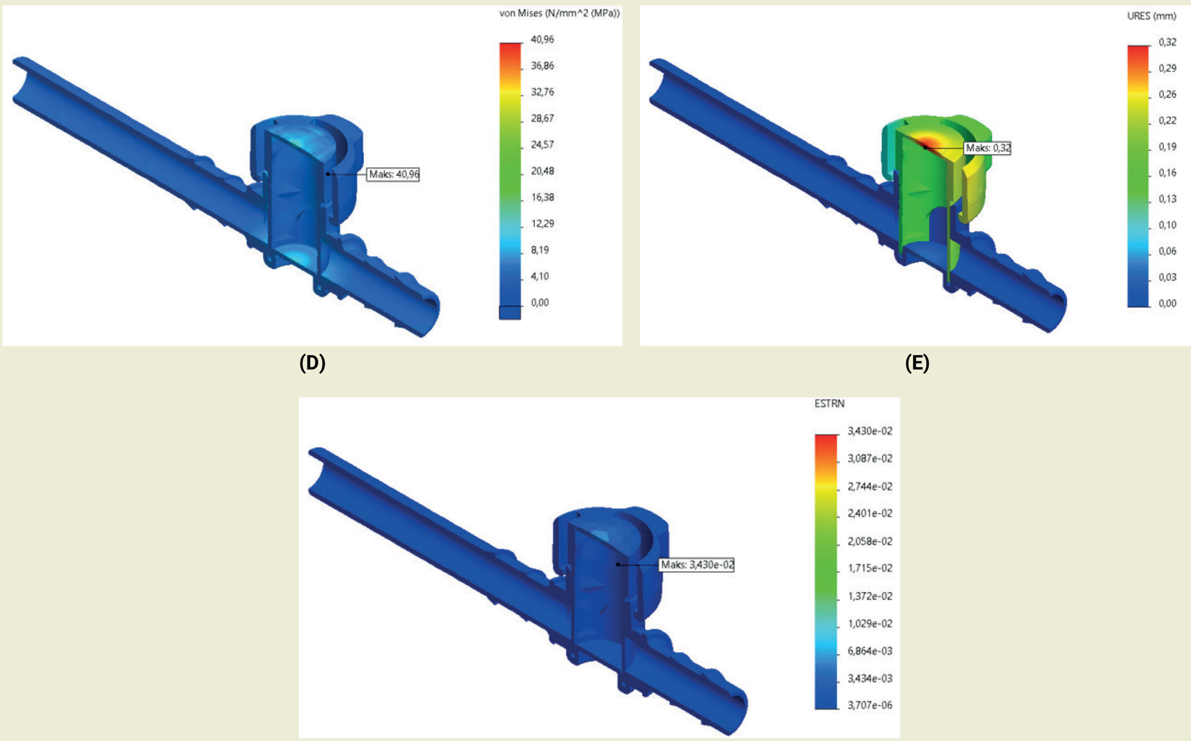
▶**Figure 6** shows the Von Mises stresses, displacements and strains experienced by the mini valve under 4 bar pressure. As can be seen from ▶**Figure 6** (A), the highest stress occurs at the edges of the plug connection between the valve body and the on-off handle. Since the flexural yield strength of polypropylene material is 44.1



(A) **(B)**

(C)

Figure 5. Pressure at mini valve with 2 bar pressure A) Von Mises Stresses B) Displacements C) Strains



(D) **(E)**

(F)

Figure 6. Formed with 4 bar pressure in the mini valve D) Stresses E) Displacements F) Strains

MPa, this stress calculated as approximately 41 MPa is below the limit value. As shown in ▶Figure 6 (B), a maximum displacement of 0.32 mm was observed in the valve at this pressure level. In addition, the highest strain occurring under a pressure of 4 bar is 3.430 x 10^{-2}, which occurs in the snap-fit fastener (▶Figure 6 (C)).

▶Figure 7 shows the Von Mises stresses, displacements and strains that the mini valve is subjected to under 6 bar pressure.

▶Figure 7 (A) shows that the highest stress occurs at the edges of the plug connection between the valve body and the on-off lever. This stress, calculated to be approximately 61 MPa, exceeds the flexural yield strength of the polypropylene material of 44.1 MPa. As shown in ▶Figure 7 (B), a maximum displacement of 0.48 mm occurred in the mini valve at this pressure level. Furthermore, the highest strain value was determined to be 5.146 x 10^{-2}, which was concentrated in the plug-in fitting (▶Figure 7 (C)).

A comprehensive table has been created that directly compares the yield and rupture stresses of POM (Acetal) and PP materials with the maximum Von Mises stresses obtained in the simulation (Table 2). When ▶Table 2 is examined, the stresses at 2 bar pressure are well below the yield limits of both POM and PP materials. No permanent deformation is expected in the material at this level. At 4 bar pressure, the Von Mises stress applied to the PP material approaches the yield stress. At 6 bar pressure, the yield stress in the PP material has been exceeded and is approaching the rupture stress. This indicates that the material is at risk of permanent deformation or fracture. It has also been determined from the table that the maximum deformation occurs for 6 bar.

3.1. Factor of Safety (FOS) Analysis and Engineering Implications

The Factor of Safety (FOS) is a critical parameter in structural design, defined as the ratio of material strength to the maximum applied stress:

$$FOS = \frac{\sigma_{yield}}{\sigma_{max}} \tag{5}$$

where σ_{yield} is the yield strength of the material and σ_{max}

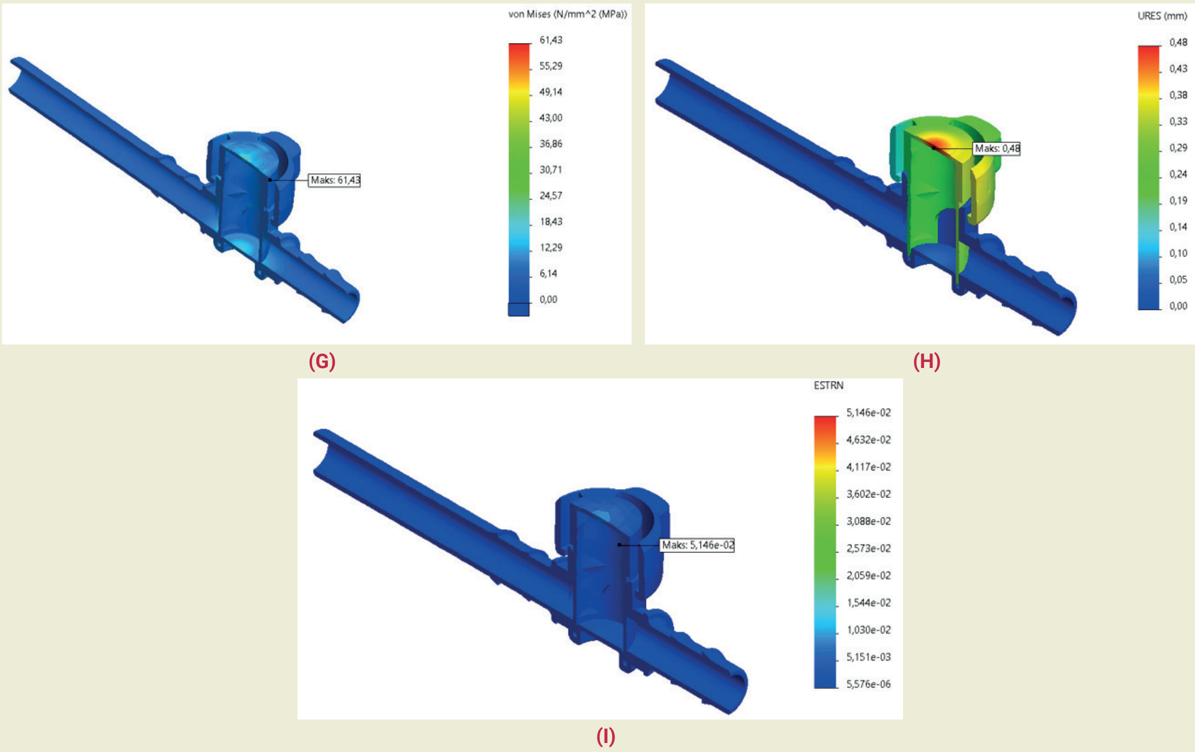


Figure 7. Formed with 6 bar pressure in the mini valve G) Stresses H) Displacements I) Strains

Table 2. Comparison of the yield and rupture stresses of POM and PP materials with the maximum Von Mises stresses obtained in the simulation.

Pressure (bar)	POM Yield Stress (MPa) [14]	POM Fracture Stress (MPa)[14]	PP Yield Stress (MPa)[15]	PP Fracture Stress (MPa)[15]	Maximum Von Mises Stress (MPa)	Maximum Displacement (mm)
2 bar	60-70	70-85	30-40	35-50	20 MPa	0.16 mm
4 bar	60-70	70-85	30-40	35-50	41 MPa	0.32 mm
6 bar	60-70	70-85	30-40	35-50	61 MPa	0.48 mm

is the maximum von Mises stress obtained from the simulation.

The results in ▶Table 3 show that:

At 2 bar, the safety factors for both materials remain well above the critical threshold (FOS > 1.5), ensuring safe operation. At 4 bar, the PP component approaches its yield limit (FOS = 0.73), indicating that it is nearing structural failure. At 6 bar, PP material falls below the safe operating range (FOS = 0.49), confirming that it will undergo plastic deformation or failure.

This analysis reinforces the operating pressure recommendations for mini plastic valves. While the POM body remains within acceptable limits, the PP handle becomes structurally unreliable above 4 bar. Therefore, limiting the valve operation to ≤ 4 bar is essential to maintain long-term mechanical integrity.

▶Figure 8 shows the Von Mises stresses in the mini valve at 2, 4 and 6 bar pressure levels.

A linear increase in Von Mises stresses is observed as the pressure increases. When 2 bar pressure is applied, the stress in the valve is well below the yield limit of the material used. However, this difference decreases as the pressure level increases and the stress value reached at 4 bar pressure is almost the same as the yield stress. At higher pressures, such as 6 bar, the stress exceeds the yield limit, straining the material strength.

▶Figure 9 shows the displacements and strains occurring in the mini valve at 2, 4 and 6 bar pressure. There is a linear increase in both displacements and strains with the increase in pressure, and the pressures are directly proportional to the amount of increase, approximately ~2 times as shown in ▶Figure 8.

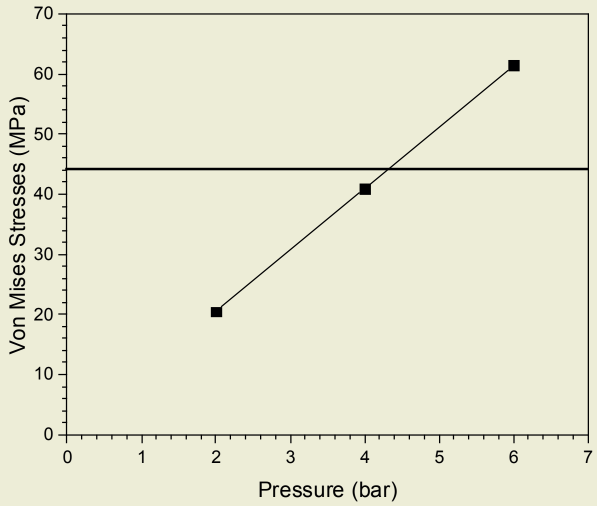


Figure 8. Von Mises stresses according to the pressures applied to the mini valve

4. Conclusions

In this study, the pressure resistance and structural integrity of a 2-way manually controlled mini plastic valve used in drip irrigation systems were analyzed using finite element analysis (FEA) under 2, 4, and 6 bar pressure conditions. The results indicate that at 2 bar,

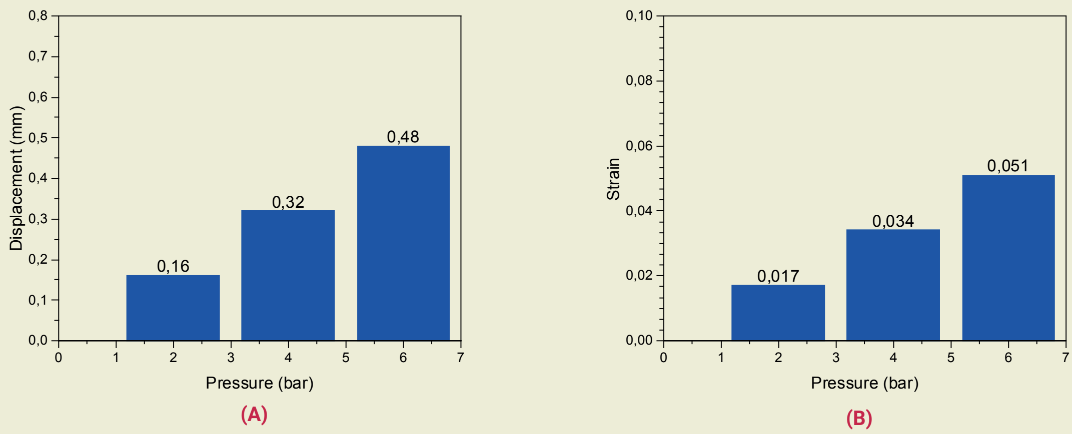


Figure 9. According to the pressures applied to the mini valve A) Displacement B) Strain

Table 3. The calculated safety factors for different pressure levels

Pressure (bar)	PP Yield Stress (MPa)	POM Yield Stress (MPa)	Max Von Mises Stress (MPa)	FOS (PP)	FOS (POM)
2 bar	30	60	20	1.50	3.00
4 bar	30	60	41	0.73	1.46
6 bar	30	60	61	0.49	0.98

 https://doi.org/10.26701/ems.1646687

the valve operates safely with a safety factor (FOS) > 1.5, while at 4 bar, the PP handle approaches its yield limit (FOS = 0.73), posing a potential structural failure risk under prolonged use. At 6 bar, the PP material exceeds its yield strength (FOS = 0.49), making it unsuitable for long-term operation due to plastic deformation or fracture risk. The study also considered hoop stress effects in inlet and outlet regions, confirming that von Mises stress evaluation inherently captures circumferential stress distribution. Based on these findings, the mini valve is recommended for safe operation at ≤ 2 bar, with 4 bar as a cautionary limit, while pressures above 4 bar should be avoided to prevent mechanical failure. It should be noted that this study only considers static pressure loading; while the results indicate that the mini valve maintains structural integrity under steady pressures, the potential impact of dynamic pressure variations, such as transient surges or cyclic loading, was not assessed. Future studies could explore these effects through transient FEA simulations, fatigue analysis, and experimental validation to provide a more comprehensive evaluation of the valve's long-term durability in real-world applications.

Research ethics

Not applicable.

Author contributions

Conceptualization: [Murat Kuru], Methodology: [Yavuz Turan], Formal Analysis: [Beyza Gizem Duman], Investigation: [Ibrahim Keles], Resources: [Hulya Alaca], Data Curation: [Ibrahim Keles], Writing - Original Draft Preparation: [Yavuz Turan], Writing - Review & Editing: [Beyza Gizem Duman], Visualization: [Hülya Alaca], Supervision: [Murat Kuru], Project Administration: [Ibrahim Keles]

Competing interests

The author(s) state(s) no conflict of interest.

Research funding

None declared.

Data availability

The raw data can be obtained on request from the corresponding author.

Peer-review

Externally peer-reviewed.

Orcid

Yavuz Turan https://orcid.org/0000-0001-8537-3989
Murat Kuru https://orcid.org/0009-0005-3283-3458
Beyza Gizem Duman https://orcid.org/0009-0004-9045-8142
Hülya Alaca https://orcid.org/0009-0004-0588-1633
İbrahim Keleş https://orcid.org/0000-0001-8252-2635

References

[1] Rajkumar, B. (2014). Globe Valve Design Optimization & Trends. L&T Integrated Engineering Services, Fluids Handling Principles and practice, Pg, 212-234.

[2] Chern, M. J., Wang, C. C., & Ma, C. H. (2007). Performance test and flow visualization of ball valve. Experimental thermal and fluid science, 31(6), 505-512. https://doi.org/10.1016/j.expthermflusci.2006.04.019

[3] Jadhav, S. S. (2014). Design of M-Type Gate Valve. International Journal of Science and Research (IJSR), 3(11).

[4] Prakash, K. S., Nazirudeen, S. M., Malvinraj, M. J., & Manohar, T. (2009). Integration of Material Design and Product Design-A CFD Based Approach. International Journal of Dynamics of Fluids, 5(2), 199-214. https://doi.org/10.52842/conf.ecaade.2015.1.665

[5] Mokhtarzadeh-Dehghan, M. R., Ladommatos, N., & Brennan, T. J. (1997). Finite element analysis of flow in a hydraulic pressure valve. Applied Mathematical Modelling, 21(7), 437-445. https://doi.org/10.1016/S0307-904X(97)00038-3

[6] Song, X. G., Wang, L., & Park, Y. C. (2009). Analysis and optimization of a butterfly valve disc. Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, 223(2), 81-89. https://doi.org/10.1243/09544089JPME236

[7] Sathishkumar, S., Hemanathan, R., Gopinath, R., & Dilipkumar, D. (2017). Design and analysis of gate valve body and seat ring. Int. J. Mech. Eng. Technol, 8(3), 131-141.

[8] Prakash, K. S., Nazirudeen, S. M., Malvinraj, M. J., & Manohar, T. (2009). Integration of Material Design and Product Design-A CFD Based Approach. International Journal of Dynamics of Fluids, 5(2), 199-214. https://doi.org/10.52842/conf.ecaade.2015.1.665

[9] Song, X., Wang, L., Kang, J., Kim, S. G., Jo, Y. J., & Park, Y. (2008, October). Optimization Of Valve Disc Using Orthogonal Array And Kriging Model. In AIP Conference Proceedings (Vol. 1052, No. 1, pp. 315-320). American Institute of Physics.

[10] Raut, L. B., & Parase, P. K. (2014). Weight Reduction of 12"-150 Class Plug valve Casting Body by Finite Element Analysis and Experimental Method. American Journal of Engineering Research, 4(4), 108-116.

[11] Lee, J. H., Song, X. G., Kang, S. M., & Park, Y. C. (2010, June). Optimization of flow coefficient for pan check valve by fluid dynamic analysis. In AIP Conference Proceedings (Vol. 1239, No. 1, pp. 337-340). American Institute of Physics.

[12] Yang, S. M., Baek, S. H., & Kang, S. (2012). Shape design for disc of a double-eccentric butterfly valve using the topology optimization technique. Journal of computational fluids engineering, 17(1), 61-69.

[13] SolidWorks Product. 2024. Nonlinear - Solution, SolidWorks Simulation Online Tutorial.

[14] Loyer, C., Ferreira, P., Marijon, J. B., Michel, V., Regnier, G., Vera, J., ... & Richaud, E. (2022). Embrittlement of polybutylene terephthalate induced by injection molding. Polymer Degradation and Stability, 196, 109843.

[15] Rösch, J., & Mülhaupt, R. (1995). Deformation behavior of polypropylene/polyamide blends. Journal of applied polymer science, 56(12), 1607-1613.