

UOT: 539

DOI: <https://doi.org/10.30546/09085.2026.01.510>

STATIC ANALYSIS OF A FULLY OPEN GATE VALVE USING THE FINITE ELEMENT METHOD IN SOLIDWORKS

N. S. RZAYEV¹, M. M. BASHIROV¹, V. F. GAHRAMANOV¹, O.H. MIRZAYEV¹

¹Baku Engineering University, Hasan Aliyev str., 120, Baku, AZ0102, Azerbaijan

nzayev@beu.edu.az

ARTICLE INFO	ABSTRACT
<i>Article history:</i> <i>Received:</i> 2025-10-23 <i>Received in revised form:</i> 2025-11-12 <i>Accepted:</i> 2025-12-11 <i>Available online:</i> 2026-06- 30	<i>In this study, the static analysis of a fully open valve gate was carried out in SolidWorks Simulation using the Finite Element Method (FEM). The main objective of the research was to determine the distribution of stress, deformation, displacement, and safety factor of the gate under operating conditions. During the modeling process, the geometric shape and boundary conditions of the gate were defined according to its actual structural characteristics.</i>
<i>Keywords:</i> <i>Finite Element Method (FEM);</i> <i>SolidWorks Simulation;</i> <i>Static Analysis;</i> <i>Valve;</i> <i>Stress.</i>	<i>A 3D model of the valve gate was created in SolidWorks, and a static analysis was performed on this model using the SolidWorks Simulation module. The mesh parameters were first defined (a fine mesh was selected), and the load corresponding to the real operating conditions of the gate was applied. As a result of the analysis, the distributed values of stress, deformation, and displacement along the surface of the gate were obtained.</i>
<i>JEL Classification:</i> C63; L60; O33	<i>The conducted analysis allowed determining the distribution characteristics of stress and deformation, providing essential data for future structural optimization, material selection, and enhancement of operational durability.</i>

2521-6376/© 2026 The Author(s). Published by Baku Engineering University.

This is an open access article under the CC BY 4.0 license (<http://creativecommons.org/licenses/by/4.0/>).

1. INTRODUCTION

In modern industry, ensuring the safe and reliable operation of pipelines requires maintaining the high structural and mechanical integrity of one of their key components — the valves. Valves serve as essential control mechanisms in pipeline systems, responsible for regulating, preventing, or completely shutting off the flow of liquids and gases. The reliability and durability of these devices primarily depend on the strength properties and material resistance of the gate (slide plate).

In industrial applications, valves typically operate under high pressure, temperature, and mechanical loads, making it especially important to study the distribution of stress and deformation in their components — particularly in the gate section. Traditional analytical methods often fail to provide accurate results under such complex geometric and loading conditions, which is why the Finite Element Method (FEM) has become widely used in modern

engineering analyses. This method enables the modeling of the behavior of complex structures under load and provides results that closely approximate real operating conditions.

In this study, the static analysis of a fully open valve gate was performed using the SolidWorks Simulation environment. During modeling, the actual dimensions and material properties of the gate were taken into account, and the distribution of stress, deformation, displacement, and safety factor across various regions of the gate was analyzed.

Traditional analytical methods do not provide accurate analysis for structures with complex geometries and non-uniform loading conditions. Therefore, over the past decades, the Finite Element Method (FEM) has become a key tool for modeling and analyzing the behavior of such mechanical systems. This method allows the discretization of complex mechanical structures into smaller elements, enabling the simulation of real loading conditions in a computer environment based on the interaction between these elements.

The fundamental principles and theoretical basis of FEM have been thoroughly developed in study (1). That research demonstrates that the finite element approach allows for precise prediction of stress and deformation distributions in various materials and under different boundary conditions. The application of FEM in the design of mechanical structures is of great importance in terms of both safety and economic efficiency. In studies (2) and (3), the stress and deformation distributions in valve gate components used in industry were analyzed using FEM, and the results showed a high correlation with experimental observations.

Modern CAD/CAE software, including SolidWorks Simulation, has made the application of this method more practical and visually interpretable. The analysis modules available in the SolidWorks environment allow for the automatic calculation of stress, deformation, and safety factor, with the results displayed as graphical distribution maps, as described in detail in study (4). In study (5), the implementation of finite element analysis in mechanical structures and its advantages in computational accuracy were demonstrated. These studies show that modeling performed using modern engineering software enables a more comprehensive evaluation of real operating conditions compared to conventional analytical calculations.

Thus, the literature review indicates that static analyses conducted using the Finite Element Method (FEM) and programs such as SolidWorks Simulation are among the most effective approaches for assessing the reliability, safety level, and optimization potential of mechanical structures.

In this study, the static analysis of a fully open valve gate was carried out using the SolidWorks Simulation module. As a result of the modeling, the distributions of stress, deformation, displacement, and safety factor were examined, and the mechanical behavior of the gate was evaluated. The findings provide a valuable basis for future research aimed at improving the reliability, shape, and material optimization of such structural components.

2. STATIC ANALYSIS OF THE GATE

The simulation of the gate was carried out in SolidWorks according to the specified parameters. For this purpose, a 3D model of the gate was created, and mesh properties were applied. The pressure value on the surface of the gate was assumed to be equal to the valve operating pressure. As a result of the simulation, the distributed values of deformation, displacement, and stress were obtained (Figure 1).

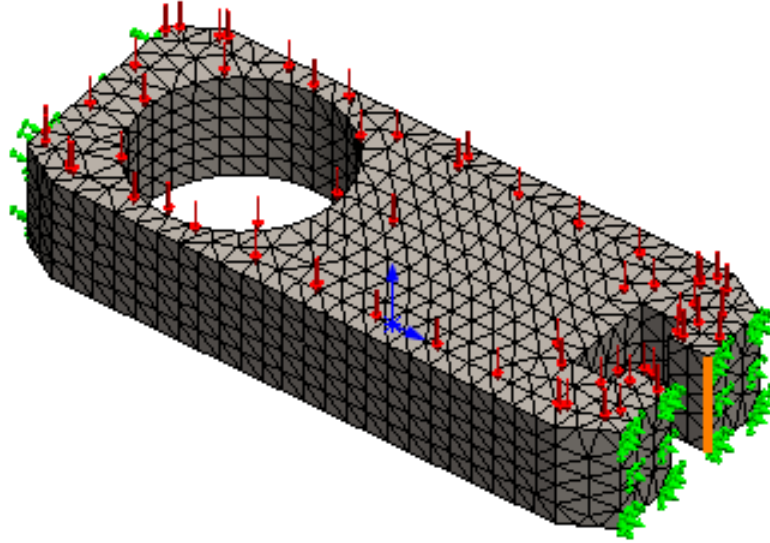


Fig. 1. Simulation of the Gate

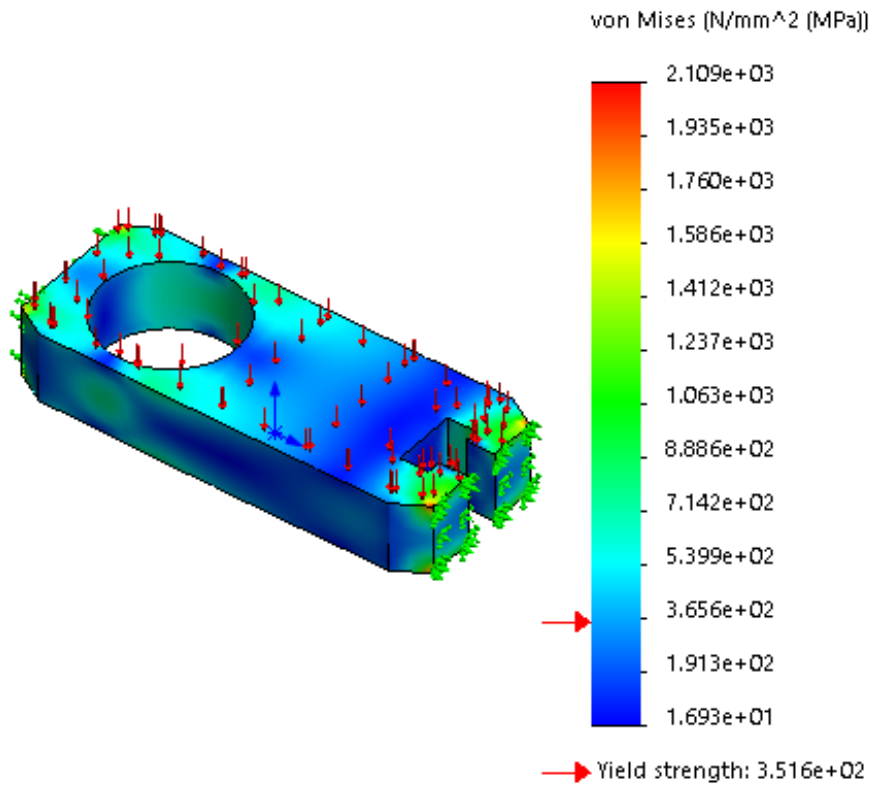
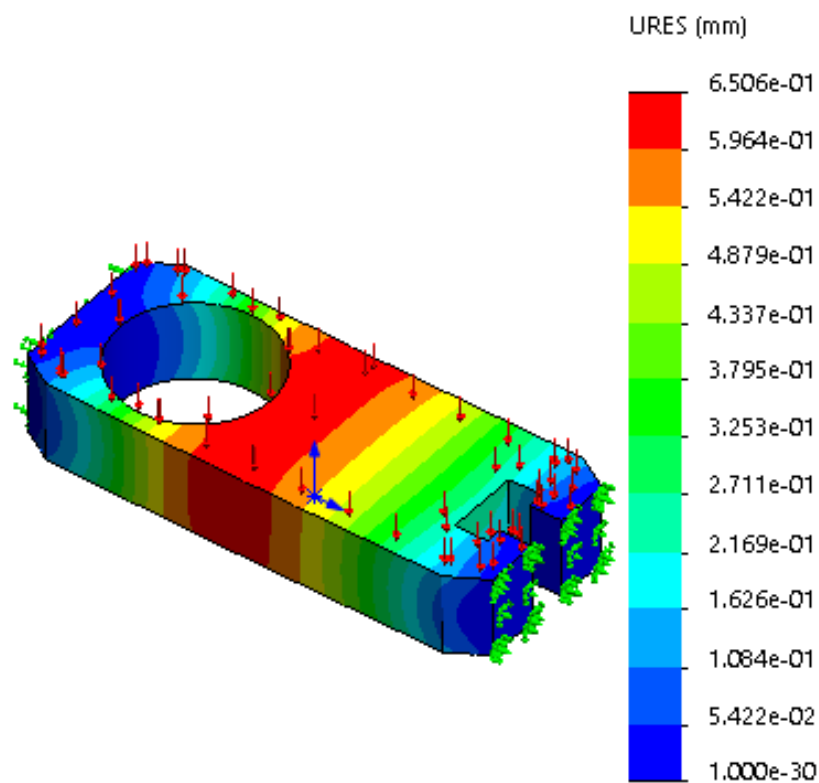
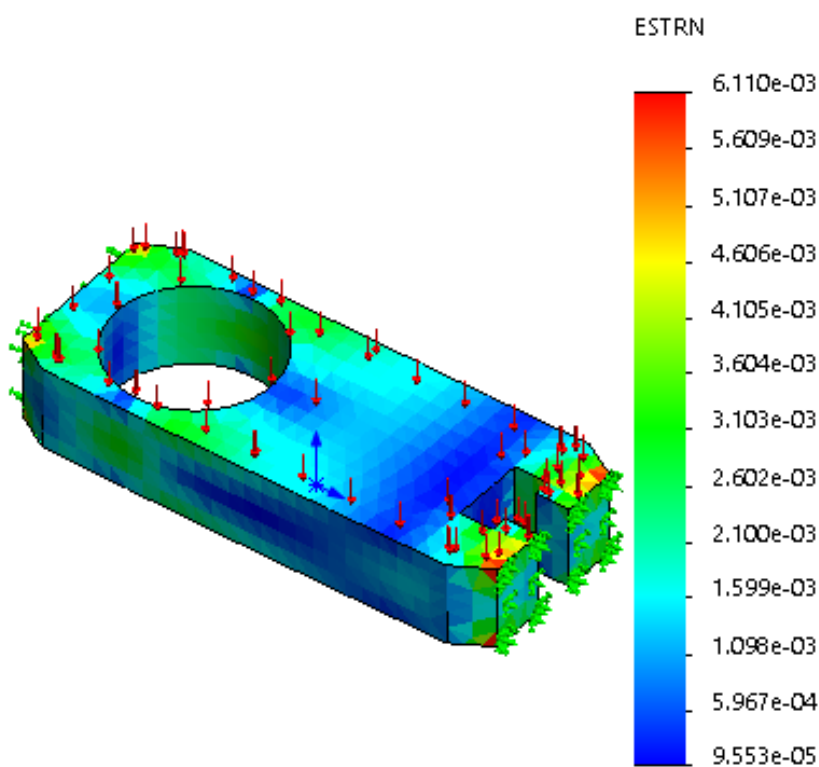


Fig. 2. Stress Distribution on the Gate Surface

The distribution of displacement on the gate surface is shown in Figure 3. As observed, the maximum displacement occurs at the central point of the gate (mid-cross-section). The highest deformation values are observed in the regions near the reinforced surfaces (Figure 3).



a)



b)

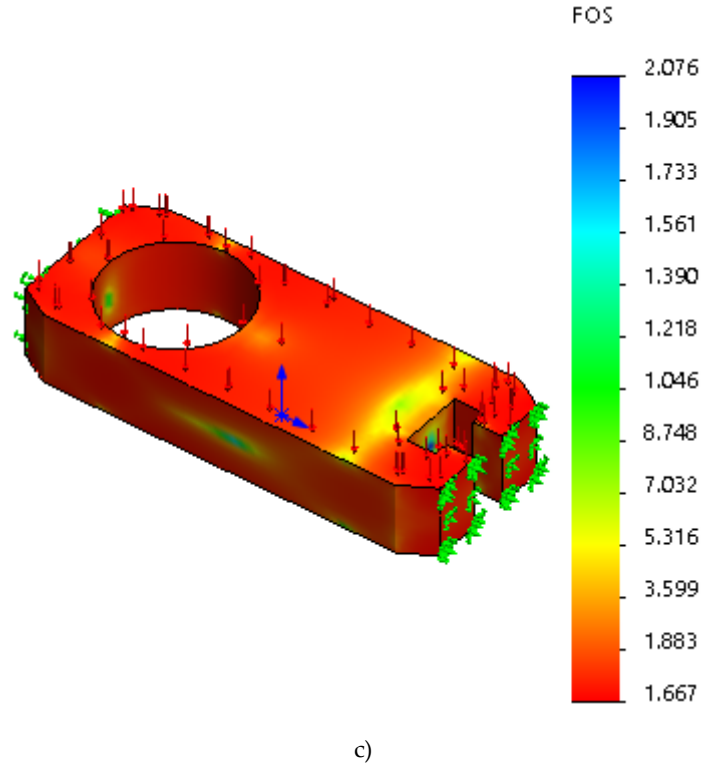


Fig. 3. Simulation Results: a) Displacement Results; b) Deformation Results; c) Safety Factor

The distribution of safety factor values is shown in Figure 5c. For brittle materials, the safety factor is calculated as the ratio of the ultimate stress to the applied working stress.

$$n = \frac{\sigma_h}{\sigma} \quad (1)$$

Here, n – is the safety factor, σ_h – is the ultimate stress, and σ – is the applied stress.

For ductile materials, the safety factor is calculated as the ratio of the yield strength to the applied stress. For the valve gate, the maximum safety factor was found to be 2, while the minimum value was 1.66. The highest safety factor values were observed near the spindle connection point, at the central surface, and at the bottom surface of the seat.

3. CONCLUSION AND DISCUSSIONS

1. Modeling based on the Finite Element Method (FEM) showed that the stress distribution in the gate is non-uniform, with the main load concentrated at the spindle contact area. This zone is potentially exposed to maximum stress and fatigue risk.
2. These distributions varied according to the stiffness and geometric characteristics of the gate, with maximum values observed on the top and bottom surfaces and minimum values at the central surface. This allows the identification of the primary elastic behavior direction and potential weak points of the gate.
3. The calculated safety factors remained within the material's elastic limit, indicating that the gate can operate reliably under static loading conditions. The analysis confirms that the material selection and structural design are appropriate.

4. Based on the results, certain modifications can be made to the gate's structure to reduce local stresses and optimize material usage. This can enhance mechanical strength and reduce manufacturing costs. These findings provide a practical basis for future studies aimed at optimizing valve components, increasing load capacity, and minimizing energy losses.
5. Overall, the analysis demonstrated that FEM-based simulation in SolidWorks is a reliable, efficient, and cost-effective tool for modern engineering evaluations.
6. It is advisable to conduct thermomechanical and dynamic analyses of the gate, as well as to test different material types and optimized design variants to further improve performance.

REFERENCES

1. Zienkiewicz, O. C., & Taylor, R. L., J.Z. Zhu. *The Finite Element Method: Its Basis and Fundamentals* (7th ed.). Elsevier, 714p, 2013.
2. Dhanwe Sunil Shivaji, & Dharmarao Sumit S. "stress analysis of gate valve by ansys", *DAV International Journal of Science*, pp.33–41. Vol.3 Issue-1, 2014.
3. Pradip Bhaskar Patil, V. R. Gambhir. "Structural Analysis of Gate Valve Body Using F.E.A" *International Journal of Engineering Research & Technology (IJERT)*, Vol. 3 Issue 6, June – 2014.
4. Tayseer Almattar. *Learn SolidWorks -2020*, 576 pg., 2019.
5. Paul M. Kurowski. *Engineering Analysis with SolidWorks Simulation*. 595 pg, 2018.
6. Hashin, Z. "Failure criteria for unidirectional fiber composites." *Journal of Applied Mechanics*, Vol 47, no 2, pp.329–334, 1980.