STRESSES AND DISPLACEMENTS IN THE MAIN SEAL OF THE SPHERICAL VALVE

Băcescu N., *Babeș-Bolyai University, Cluj-Napoca, ROMANIA*
Praisach Z.I., *Babeș-Bolyai University, Cluj-Napoca, ROMANIA*

ABSTRACT: The paper presents analytical and numerical investigation of stress and displacement behavior in the main seal of spherical valves operating under water pressure conditions typical of hydroelectric turbine systems. Finite element analysis (FEA) is employed to evaluate the mechanical response of the sealing element under steady-state hydraulic loading, with particular focus on identifying stress concentrations and deformation zones that may affect sealing performance. The influence of seal geometry and material properties is examined to determine their contribution to maintaining structural integrity and leak-tightness. The results are particularly relevant for large spherical valves used in hydroelectric power plants, where reliable sealing under high-pressure water conditions is essential to ensure operational safety and efficiency. The findings provide valuable insights into the optimization of sealing systems, supporting advanced design strategies and improved maintenance practices in energy applications.

KEY WORDS: spherical valve, finite element analysis, stress distribution, sealing performance.

1. INTRODUCTION

In hydraulic and industrial energy systems, spherical valves are key components, enabling precise management of high-pressure fluid flows and ensuring reliable operational control [1]. Recognized for their robust structure and compact execution, these valves are integral to installations where demanding performance and durability criteria must be met, such as hydroelectric power plants and complex industrial processes [2,3]. The sealing system plays a decisive role in determining valve efficiency and durability, as it must preserve leak-tightness under fluctuating and often critical conditions. Insufficient evaluation of mechanical stresses or lack of optimization can result in accelerated wear, plastic deformation, or premature failure of the sealing elements [4–6]. To address these challenges, this study employs Hertzian contact theory to analyze stress distribution and elastic deformation at the sealing interface between the seat ring and valve seat, providing a precise framework to optimize the design and enhance the durability of spherical valves in high-pressure applications [7]. The analytical results are compared and evaluated with the values obtained from the numerical static analysis using SolidWorks software.

2. ANALYTICAL MODEL

The sealing surface of the movable ring is conical, while the sealing seat has a toroidal shape. When water pressure acts on the movable ring, stresses and deformations occur at the point of contact between the two surfaces. This interaction ensures the spherical valve seals through a very narrow contact area. To estimate the maximum contact stress (Fig. 2) between the movable ring and the sealing seat, a linear contact model is used, following the approach described in [8].

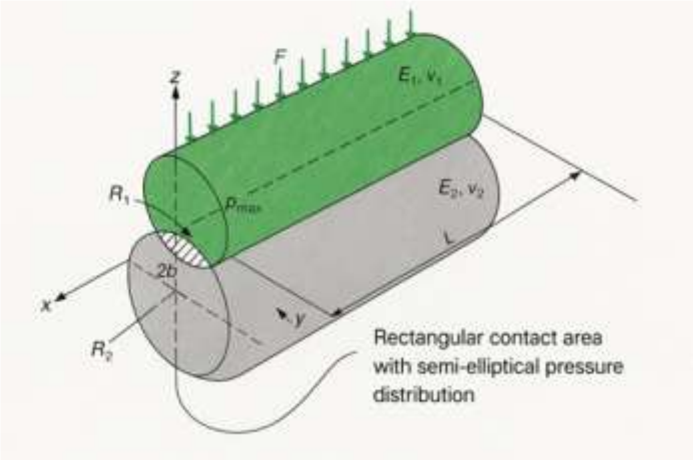

Figure 1. Diagram for stress distribution [8]

Both components are made from martensitic stainless steel and share the following properties: modulus of elasticity $E_1 = E_2 = 2{,}15 \times 10^{11}$ N/m², material density $\rho = 7800$ kg/m³. and Poisson's ratio $\nu_1 = \nu_2 = 0.28$. The system is subjected to a pressure p= 3.92266 N/mm^2. The sealing diameter is D_{et} =1.282 m, and the corresponding contact length is $L=\pi \times D_{et}$ = 4.0275 m. The force applied to the contact area is F= 52135.375 N.

The half-width of the contact area is expressed by relation (1).

The maximum contact stress $\sigma_{H\ max}$ (2) is determined using Hertzian contact theory [8] considering the external contact between a flat surface (the movable ring, $r_1 = \infty$) and a curved surface (the sealing seat, r_2=0.020 m).

$$b = \sqrt{\frac{4F}{\pi L} \cdot \frac{\frac{1-\nu_1^2}{E_1} + \frac{\nu_2^2}{E_2}}{\frac{1}{\rho_2}}} = 0{,}053789 \text{ mm} \qquad (1)$$

$$\sigma_{Hmax} = \frac{2}{\pi} \cdot \frac{F}{bL} = 153{,}2078 \text{ MPa} \qquad (2)$$

3. FINITE ELEMENT ANALYSIS

To ensure the validation of the analytical results and to investigate the complex stress distribution within the contact zone, a finite element analysis (FEA) is proposed. The 3D model (Figure 2) will include both the movable ring and the valve sealing seat, incorporating the specific mechanical properties of the materials. The geometry will be defined according to the parameters detailed in the previous chapter. The sealing seat of the spherical valve was constrained, while an axial pressure of p=3.92266 MPa was applied to the movable ring, as illustrated in Figure 3.

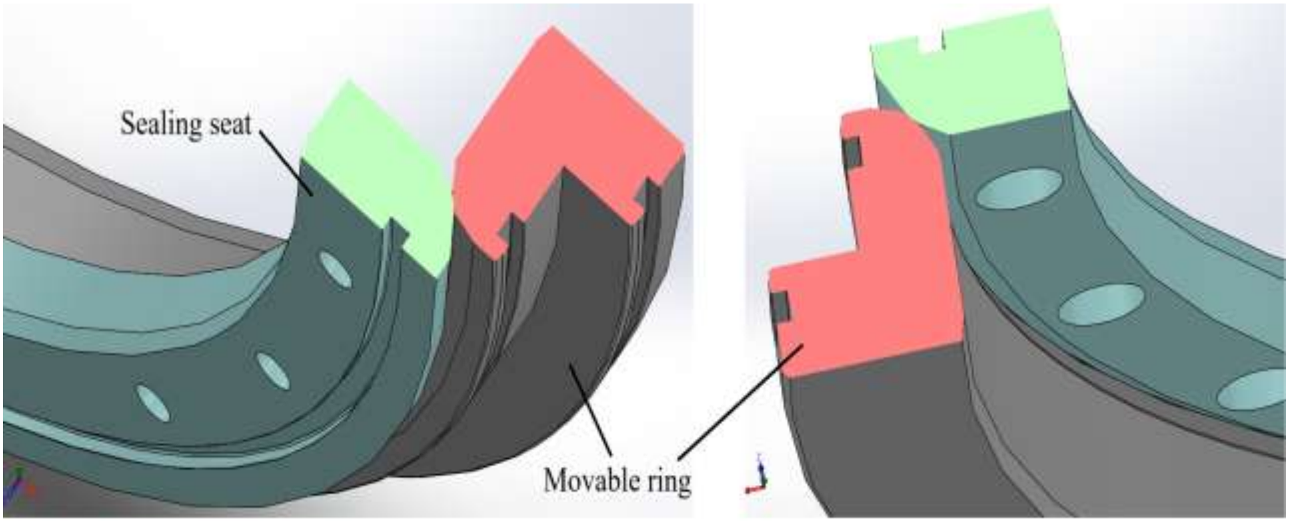

Figure 2. 3D model of the sealing system

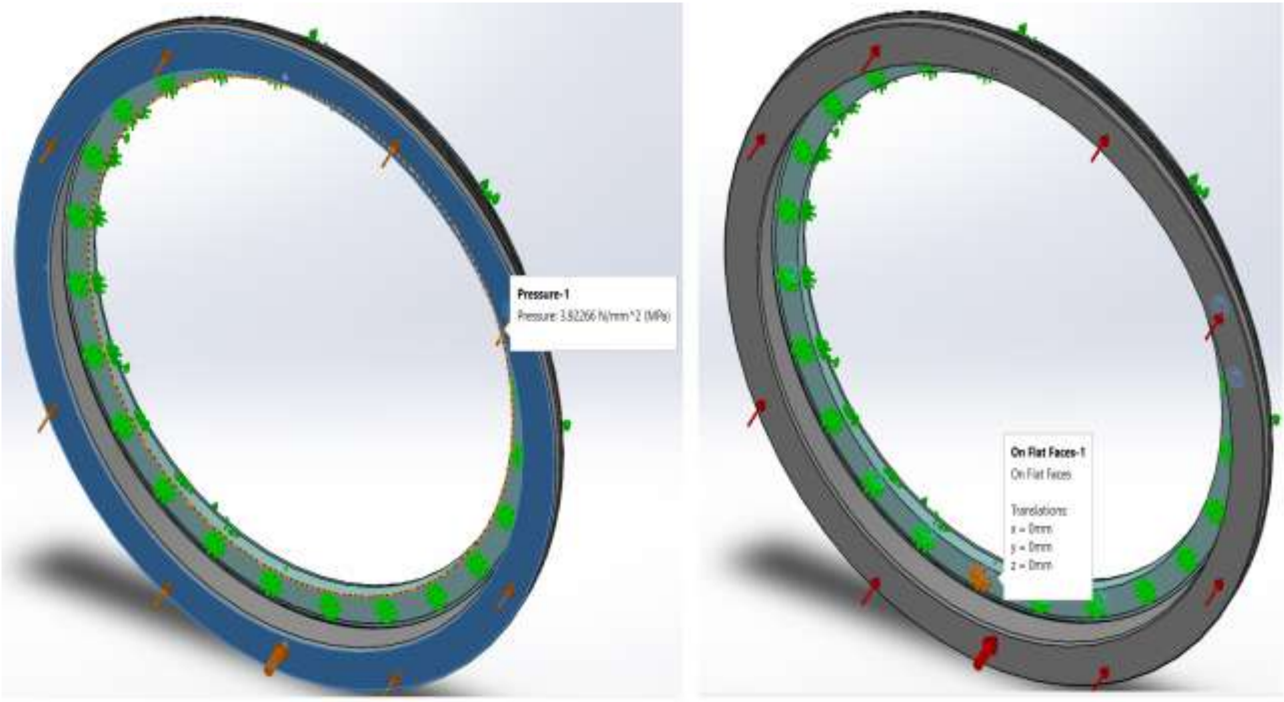

Figure 3. Axial load and fixture locations on the spherical valve sealing system

After the assembly was meshed, a blended curvature-based mesh strategy was implemented, utilizing Jacobian point control to achieve high-quality refinement in critical nodal regions. The resulting mesh configuration is illustrated in Figure 4.

Mesh Details

Study name	Static 1* (-Default-)
DetailsMesh type	Solid Mesh
Mesher Used	Blended curvature-based mesh
Jacobian points for High quality mesh	16 points
Max Element Size	33.3738 mm
Min Element Size	1.66869 mm
Mesh quality	High
Total nodes	3231538
Total elements	2109243
Maximum Aspect Ratio	20.141
Percentage of elements with Aspect Ratio < 3	97.5
Percentage of elements with Aspect Ratio > 10	0.0307
Percentage of distorted elements	0
Number of distorted elements	0
Remesh failed parts independently	Off
Reuse mesh for identical bodies	Off
Number of bodies that have reused mesh	0
Time to complete mesh(hh:mm:ss)	00:01:35
Computer name	

Figure 4. Mesh details.

The primary objective of this analysis is to validate the analytically established value of the maximum contact stress, s_{Hmax} (2), by comparison with the static results generated through finite element modeling. The finite element method revealed a peak axial compressive stress of -157.4 MPa, represented in Figure 5. In the analytical calculation, a value of 153.2078 MPa was obtained, which is close to the one computed using the finite element method, with a deviation of 2.663%. The discrepancy between the positive and negative values can be explained by reporting conventions: in the analytical approach, the contact stress is presented as an absolute (positive) value, whereas the finite element analysis reveals both the magnitude and the direction, with the negative sign indicating the compressive nature of the stress within the contact area.

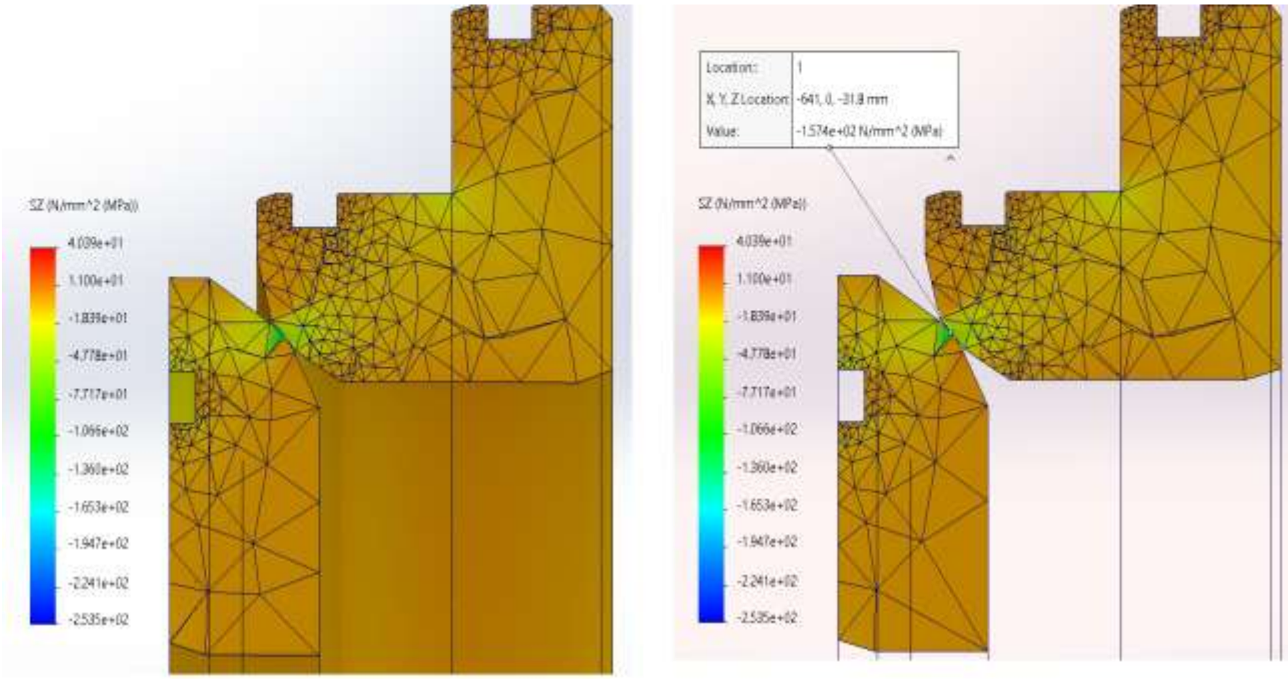

Figure 5. Static nodal stress distribution in the axial direction

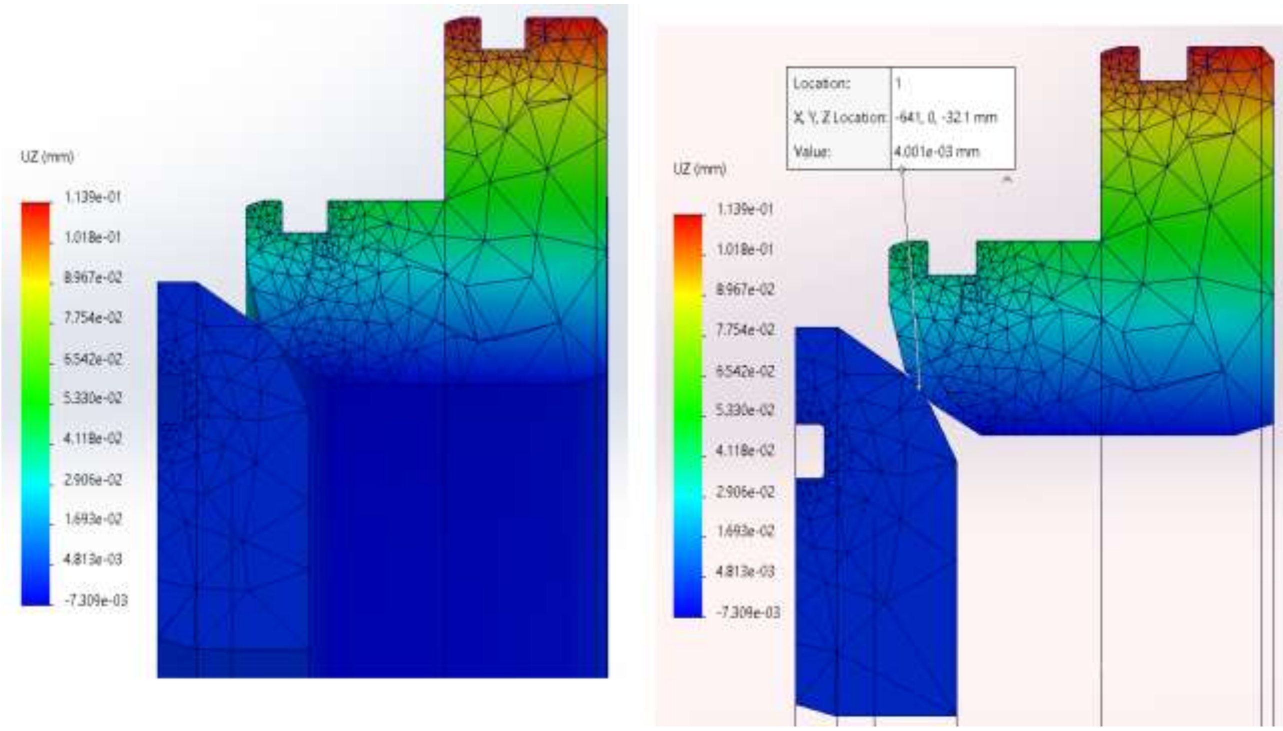

Figure 6. Static displacements in the axial direction

Figure 6 illustrates the static axial displacements, with the maximum observed on the outer surface of the movable ring, measuring 0.0116 mm. Within the contact region between the two components, the displacement reaches 0.004 mm, corresponding to the compression of the surface roughness because of the applied loading.

5. CONCLUSION

This study focuses on evaluating the contact stress between the sealing surfaces of a high-performance spherical valve by employing two distinct assessment methodologies. The close alignment between theoretical estimations and finite element simulations confirms the reliability of the modeling approach used to assess contact stress in sealing components. The numerical analysis not only supports the validity of the method but also provides enhanced insight into localized stress zones, which are crucial for refining the design and ensuring consistent performance over time. A deviation of just 2.663% between the two methods highlights the continued effectiveness of simplified analytical models, such as Hertzian contact theory, for preliminary design and validation purposes. Analytical methods offer a fast way to estimate maximum stress and contact area dimensions. In contrast, numerical static analysis, while more resource-intensive in terms of time, hardware, and software, enables a broader understanding of stress and deformation behavior throughout the entire assembly, not just at the contact interface. The findings of this study contribute to a deeper understanding of sealing system mechanics and offer valuable guidance for future design improvements, particularly in applications requiring high precision and structural resilience under pressure.

Furthermore, integrating both analytical and numerical approaches throughout the design process enables efficient preliminary validation and informs subsequent detailed analysis, ultimately optimizing performance and reliability.

REFERENCES

[1] White, F, Fluid Mechanics, McGraw-Hill Higher. Education, 862 pg., 2011

[2] Huang, X. Yan, R. Larsson, and A. Almqvist, "Numerical Simulation of Static Seal Con-tact Mechanics Including Hydrostatic Load at the Contacting Interface," Lubricants, vol. 8, no. 2, p. 22, 2020

[3] Anwar A., Gorash Y., Dempster W., Application of Multiscale Approaches to the Investigation of Sealing Surface Deformation for the Improvement of Leak Tightness in Pressure Relief Valves, Proc. ASME Pressure Vessels and Piping Conf., (2016)

[4] Todhunter IA. History of the Theory of Elasticity and of the Strength of Materials. New York: Cambridge University Press;2014

[5] Lucci B. N., Lamas W., Grandinetti F. J., Giacaglia G., Finite Element Analysis Applied to a Metal-to-Metal Seal Design, in Proc. 26th Int. Congr. of Mechanical Engineering (COBEM), (2020)

[6] Timoshenko, S., Goodier, J., Theory of Elasticity, third ed. McGraw-Hill, New York, (1970)

[7] Houpert, L., An Engineering Approach to Hertzian Contact Elasticity - Part I and II. ASME J. Tribology 123, (2001).

[8] X. Qin, "Contact Strength of Material," in Strength of Materials, Intech Open, Mar., (2020).