Serbian Tribology Society

SERBIATRIB '25

19th International Conference on Tribology

Faculty of Engineering
University of Kragujevac

Kragujevac, Serbia, 14 – 16 May 2025

Research paper DOI:10.24874/ST.25.163

FINITE ELEMENT ANALYSIS OF STRESS AND CONTACT PRESSURE IN STEEL PLATES UNDER VARYING FRICTION COEFFICIENTS

Vladimir MILOVANOVIC[1], Milos PESIC[2,*], Snezana VULOVIC[2], Zivana JOVANOVIC PESIC[1], Miroslav ZIVKOVIC[1]

[1]University of Kragujevac, Faculty of Engineering, Kragujevac, Republic of Serbia
[2]University of Kragujevac, Institute for Information Technologies - National Institute of the Republic of Serbia, Kragujevac, Republic of Serbia
*Corresponding author: milospesic@uni.kg.ac.rs

Abstract: *This study investigates the impact of the friction coefficient on stress distribution and contact pressure in numerical simulations of contact between two plates under vertical loading and specified displacement. Using the finite element method (FEM), the plates are analysed with friction coefficients varying from 0.1 to 0.9. The analysis includes three different mesh densities and two types of finite elements to assess the influence of mesh refinement and element type on the accuracy of the results. The results indicate that friction significantly affects the stress distribution and contact pressure, with notable differences observed as the friction coefficient increases. The study highlights the importance of selecting appropriate mesh density and element type to ensure the accuracy of the simulations, contributing to a better understanding of frictional contact in engineering applications. The findings are valuable for optimizing material design and contact mechanics in practical applications, where accurate predictions of material behaviour are essential for performance and durability.*

Keywords: *friction coefficient, finite element method, contact mechanics, mesh density, element type*

1. INTRODUCTION

Contact problems are fundamental in engineering analyses, as they frequently arise in a wide range of applications, from structural engineering to biomechanical simulations. In the context of numerical simulations, understanding and accurately modelling the interaction between contacting bodies is crucial for ensuring the precision of results, particularly in scenarios involving friction. The friction coefficient, which dictates the interaction between two surfaces in contact, significantly influences the stress distribution and contact pressure, directly impacting the behaviour of a system under load [1, 2].

Finite element methods (FEM) provide a powerful approach for solving contact problems, enabling precise analysis of stress and pressure distribution at contact points, as well as material behaviour under various loading conditions. However, the choice of mesh density and element type plays a critical role in the accuracy of these simulations. Fine meshes and appropriate element types allow for a more accurate representation of local contact effects, while overly coarse meshes may lead to a loss of precision [3, 4].

This study investigates the impact of the friction coefficient on stress distribution and contact pressure in numerical analyses of contact between two plates. The plates are subjected to vertical loading and a specified displacement, with friction coefficients varied from 0 to 1. The analysis is performed using three different mesh densities and two types of finite elements, providing a detailed assessment of how these parameters influence simulation results. Such studies are crucial for understanding the role of friction in various engineering applications, where accurate predictions of material behaviour are essential for design optimization [5].

The findings of this study contribute to a better understanding of the role of friction in contact mechanics and the importance of mesh and element selection for the accuracy of numerical models. Further analysis will demonstrate how variations in these parameters affect material behaviour, which is critical for optimizing engineering designs in practical applications [6].

2. METHODOLOGY

This study aims to investigate the influence of the friction coefficient on stress distribution and contact pressure between two plates using a static finite element analysis. The plates are assumed to be rigid, and the material behaviour is treated as linear elastic. The following sections outline the details of the numerical model, including material properties, mesh configuration, and boundary conditions applied in the simulations.

2.1 Geometry and Material Properties

The numerical analysis was performed on two plates with dimensions of 120 mm × 60 mm × 5 mm. The plates are modelled as a rigid steel material subjected to a vertical load of 10 kN and a displacement of 30 mm imposed in the perpendicular direction. The material properties used for the simulations are as follows: Young's Modulus – 2.1 10^5 MPa, Poisson's Ratio - 0.3 and Density - 7.85 10^{-9} t/mm^3.

The steel material is assumed to exhibit linear elastic behavior under the applied load.

2.2 Friction Model

The friction between the two plates is modeled using the Coulomb friction law, where the friction force F_f is calculated as:

$$F_f = \mu \cdot N \quad (1)$$

where μ is the friction coefficient, and N is the normal contact force. The friction coefficient μ is varied from 0.1 to 0.9 in increments of 0.1 to examine its influence on the stress distribution and contact pressure.

2.3 Finite Element Model

The numerical simulations were carried out using the finite element method (FEM), employing both 3D hexahedral elements and 2D plate elements to study the friction coefficient variations in contact region. Both models represent the same physical scenario, with variations in mesh density and element type. The plates are subjected to a vertical force of 10 kN, applied uniformly over the contact region on top surface of the upper plate. A displacement of 30 mm is imposed along the perpendicular direction of the plates. The outer edges of the lower plate are fixed in all directions as shown in Figure 1.

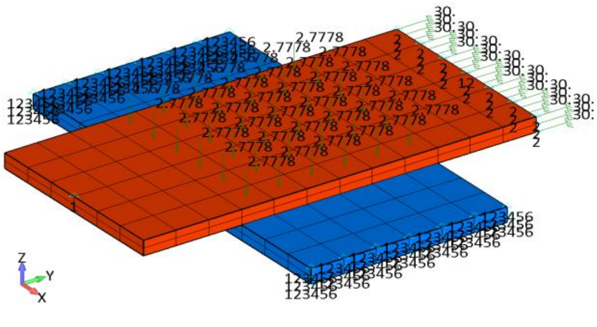

Figure 1. Geometry model with given loads and constraints

3D hexahedral elements were used to provide a detailed representation of the material behaviour and contact interaction, while 2D plate elements were used because of their more efficient computation.

Both FE models use three different mesh densities to evaluate the effect of mesh refinement: coarse mesh with 20 mm x 20 mm x 5 mm for plate elements, 20 mm x 20 mm x 2.5 mm for 3D hexahedral elements, meduim mesh with 10 mm x 10 mm x 5 mm for plate elements, 10 mm x 10 mm x 2.5 mm for 3D hexahedral elements and fine mesh with 5 mm x 5 mm x 5 mm for plate elements and 5 mm x 5 mm x 2.5 mm for 3D hexahedral elements.

The mesh density is varied in both the 3D and 2D models to assess the impact of mesh refinement.

The numerical simulations are performed using FEMAP v2021.2 [7], a pre- and post-processor for finite element analysis.

3. RESULTS AND DISCUSSION

As already mentioned, this numerical analysis explores the impact of friction coefficient on stress distribution and contact pressure across two plate models using different mesh sizes and element types (3D hexahedral elements and 2D plate elements). The mesh sizes evaluated are 5x5x2.5 mm, 10x10x2.5 mm, and 20x20x2.5 mm for 3D hexahedral elements and 5x5x5 mm, 10x10x5 mm, and 20x20x5 mm for plate elements, which allow for an examination of how mesh refinement influences the results.

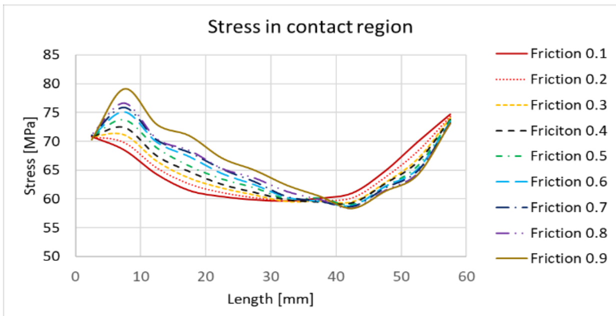

Figure 2. Stress distribution in the contact region – FE model with 3D elements 5 mm x 5 mm x 2.5 mm

Figure 2 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 3D hexahedral finite elements sized 5x5x2.5 mm, which highlight the detailed stress variations across the interface.

Figure 3 presents the contact pressure distribution in the contact region for the same FE model.

As can be seen from the Figure 2 the most significant fluctuation in stress values are as the friction coefficient increased, with noticeable peaks at lower friction coefficients that gradually stabilized.

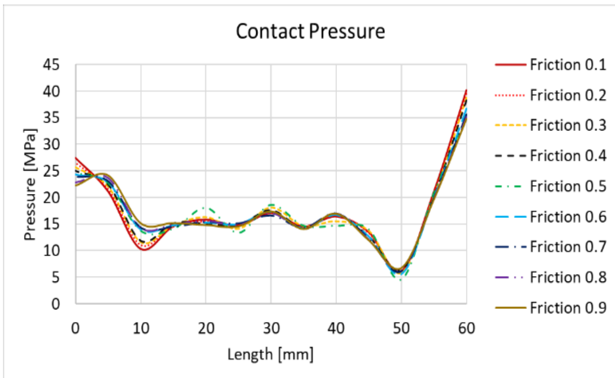

Figure 3. Contact pressure distribution in the contact region - FE model with 3D elements 5 mm x 5 mm x 2.5 mm

From the Figure 3 can be seen that the contact pressure curves displayed distinct peaks, especially at lower friction coefficients, indicating a pronounced local response to contact interactions.

Figure 4 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 3D hexahedral finite elements sized 10x10x2.5 mm, which highlight the detailed stress variations across the interface. Figure 5 presents the contact pressure distribution in the contact region for the same FE model.

In Figure 4, the diagram represents that the medium mesh size smoothed out some of the more extreme variations seen in the finer mesh, but still maintained enough detail to effectively map significant stress patterns. It showed a balanced response to different friction coefficients.

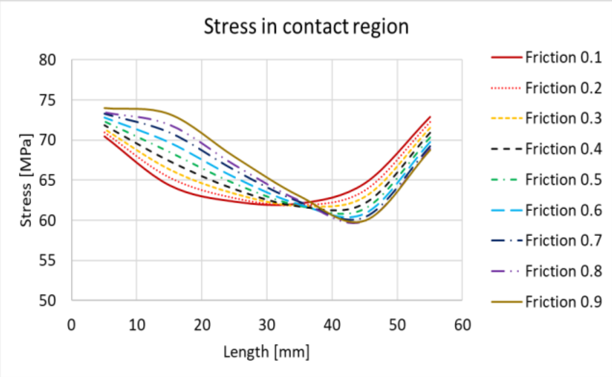

Figure 4. Stress distribution in the contact region - FE model with 3D elements 10 mm x 10 mm x 2.5 mm

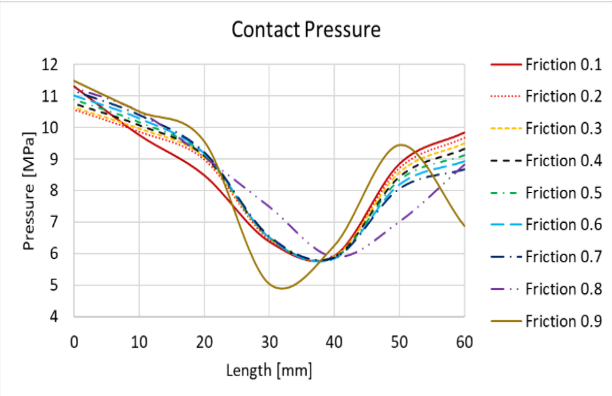

Figure 5. Contact pressure distribution in the contact region - FE model with 3D elements 10 mm x 10 mm x 2.5 mm

With a medium mesh size, the contact pressure responses were less peaked and more smoothed out compared to the finest mesh. Although the variations across different friction coefficients were moderate—indicating a good balance between detail and computational efficiency—variations across the contact region were more pronounced, as can be seen on Figure 5.

Figure 6 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 3D hexahedral finite elements sized 20x20x2.5 mm, which highlight the detailed stress variations across the interface. Figure 7 presents the contact pressure distribution in the contact region for the same FE model.

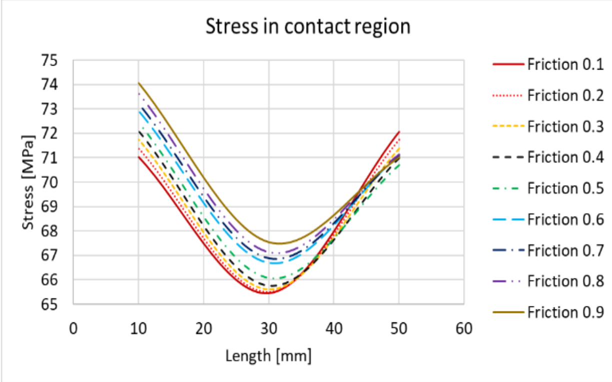

Figure 6. Stress distribution in the contact region - FE model with 3D elements 20 mm x 20 mm x 2.5 mm

As can be seem from the Figure 6 the coarsest mesh provided a broad overview of stress distribution, with much less detail and lower sensitivity to changes in friction. The stress distribution was the most uniform among the meshes, showing only major trends and omitting finer stress nuances.

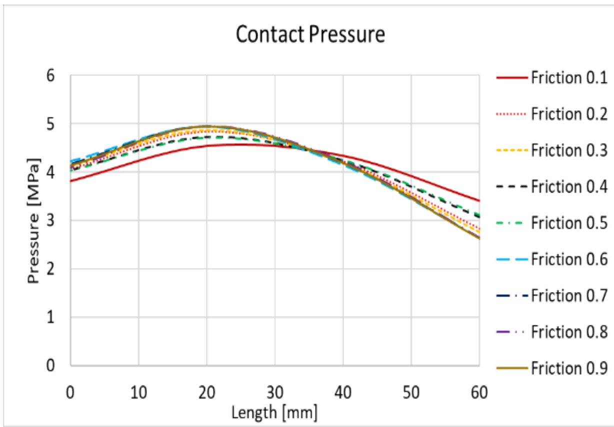

Figure 7. Contact pressure distribution in the contact region - FE model with 3D elements 20 mm x 20 mm x 2.5 mm

The coarsest mesh displayed the least sensitivity to friction changes. The pressure curves were the most smoothed and showed minimal fluctuation across different friction coefficients, as shown on the diagram on Figure 7.

Figure 8 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 2D plate elements sized 5x5x5 mm, which highlight the detailed stress variations across the interface. Figure 9 presents the contact pressure distribution in the contact region for the same FE model.

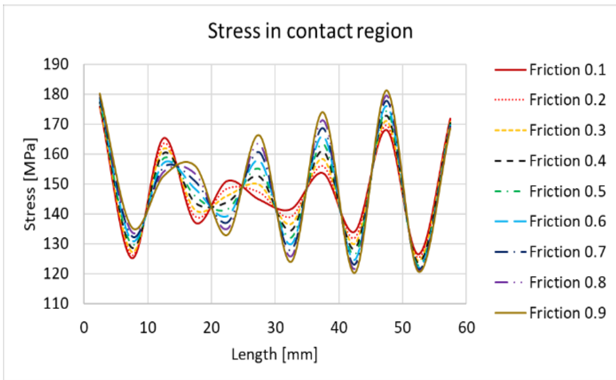

Figure 8. Stress distribution in the contact region - FE model with plate elements
5 mm x 5 mm x 5 mm

The finest mesh resolution in shell elements showed very detailed and sensitive stress responses to changes in friction coefficients. This mesh captured high peaks and sharp fluctuations, especially noticeable at lower friction coefficients, as shown in the diagram within Figure 8.

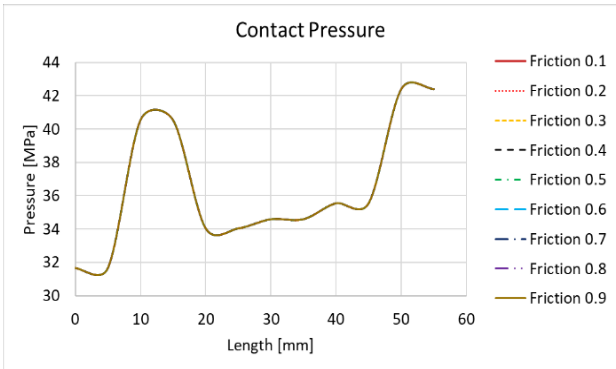

Figure 9. Contact pressure distribution in the contact region - FE model with plate elements
5 mm x 5 mm x 5 mm

Similar to the fine 3D mesh, the finest shell mesh showed high sensitivity to friction changes, with sharp variations in contact pressure across the length of the contact surface. With the finest mesh contact pressure curves are identical for all friction coefficients, as can be seen from the Figure 9.

Figure 10 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 2D plate elements sized 10x10x5 mm, which highlight the detailed stress variations across the interface. Figure 11 presents the contact pressure distribution in the contact region for the same FE model.

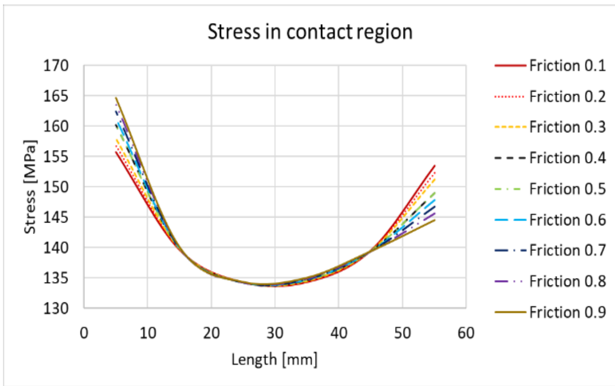

Figure 10. Stress distribution in the contact region - FE model with plate elements
10 mm x 10 mm x 5 mm

The medium mesh size displayed more smoothed stress curves than the finest mesh, reducing the visibility of extreme stress concentrations but still providing adequate detail to observe significant trends. The response to different friction coefficients was less dramatic but clearly discernible, as shown in Figure 10.

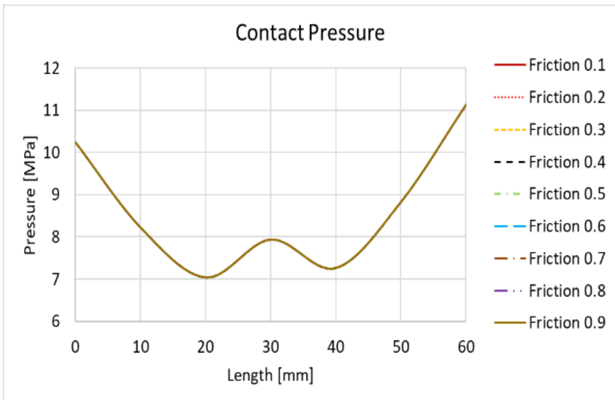

Figure 11. Contact pressure distribution in the contact region - FE model with plate elements
10 mm x 10 mm x 5 mm

Medium mesh sizes exhibited smoother pressure distributions compared to the finest mesh, with less pronounced peaks and troughs. The contact pressure curves are more uniform across different friction coefficients, showing a moderate sensitivity to changes in friction, and as for the previous FE model curves are also identical as shown in Figure 11.

Figure 12 presents the stress distribution diagram in the contact region, illustrating the results obtained from a finite element analysis conducted using 2D plate elements sized 20x20x5 mm, which highlight the detailed stress variations across the interface. Figure 13 presents the contact pressure distribution in the contact region for the same FE model.

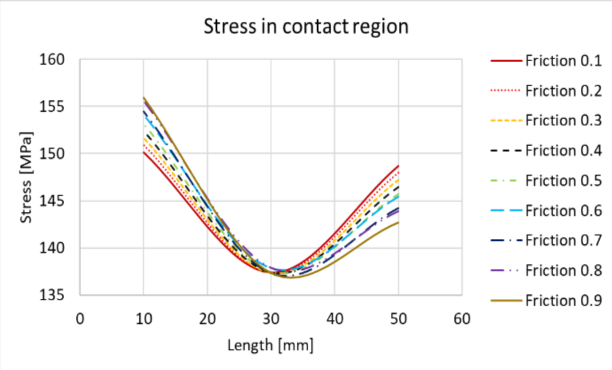

Figure 12. Stress distribution in the contact region - FE model with plate elements 20 mm x 20 mm x 5 mm

The coarsest mesh showed the least detailed stress distribution, with much smoother curves and fewer fluctuations. The impact of different friction coefficients on stress distribution was the most damped in this mesh size, indicating a broad overview rather than detailed analysis.

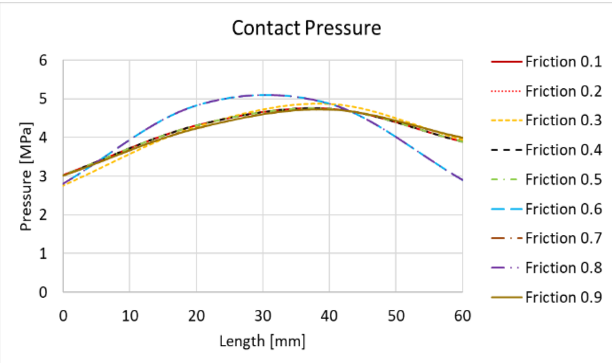

Figure 13. Contact pressure distribution in the contact region - FE model with plate elements 20 mm x 20 mm x 5 mm

The coarsest shell mesh showed the least sensitivity to changes in friction, with the smoothest pressure curves and the least variation between different friction coefficients. The overall contact pressure trends were maintained, but fine details were lost.

The stress distribution across the models demonstrated a clear dependency on the friction coefficient. At lower friction coefficients, the stress concentrations were higher and more localized, particularly near the edges of the contact area. This localization reflects the greater relative movement between surfaces, leading to higher stress peaks. As friction coefficients increased, the stresses became more distributed throughout the material, indicating that the increased friction helps to stabilize the interaction between the surfaces, thereby spreading the load more evenly and potentially improving the overall structural integrity of the assembly.

Across all mesh sizes and element types, contact pressure generally showed a tendency to be higher at lower friction coefficients, with sharp peaks indicating significant localized forces at the contact interface. As the friction coefficient increased, the overall contact pressure tended to distribute more evenly across the contact surface, resulting in smoother curves and reduced peaks. This trend suggests that higher friction levels facilitate a more uniform distribution of forces, potentially reducing the risk of excessive local stress and wear.

In the context of this study on the impact of friction coefficients on stress distribution and contact pressure in steel plates, the most commonly reported friction coefficient in the literature for steel-to-steel contact under typical conditions is approximately 0.3. The friction coefficient value, as noted in the referenced study, underscores the influence of external factors such as surface finish and lubrication, which are vital for the precise modelling of mechanical interactions in engineering applications [8].

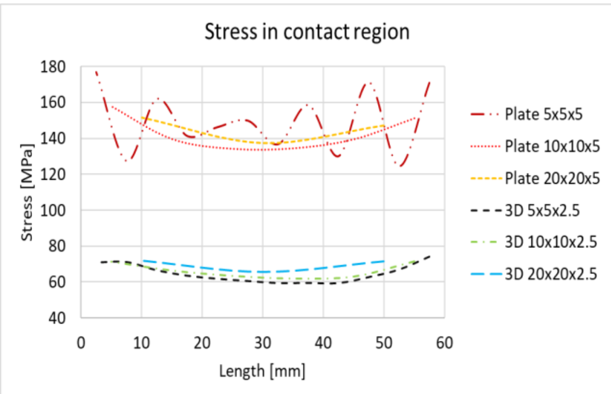

Figure 14. Stress distribution in the contact region for 0.3 friction coefficient value

To visually represent how this friction coefficient influences mechanical responses Figure 14 presents the stress distribution results with 0.3 friction coefficient value, showing how the stress varies by using different type of finite elements and different mesh sizes.

The 3D elements tend to show a more uniform and less fluctuating stress distribution compared to plate elements, especially at coarser mesh sizes (20x20x5). This could be indicative of 3D elements ability to model stress more homogeneously across the volume of the material.

The stress levels for 3D elements are notably lower than those for plate elements, particularly noticeable in the finer meshes. This might be attributed to the different ways in which plate and 3D elements handle stress distribution and transmission through the material.

Figure 15 presents the contact pressure distribution results with 0.3 friction coefficient value, showing how the contact pressure varies by using different type of finite elements and different mesh sizes.

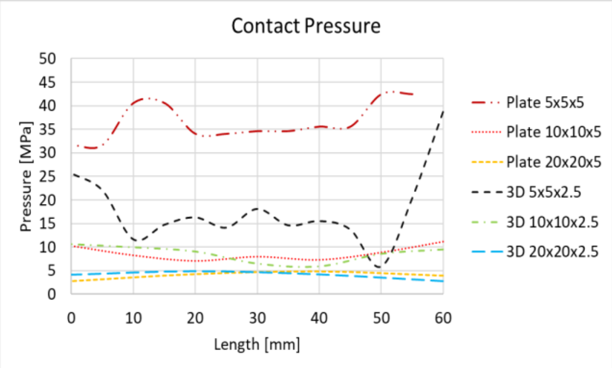

Figure 15. Contact pressure distribution in the contact region for 0.3 friction coefficient value

The 3D elements generally exhibit more stable and consistent contact pressure distributions compared to plate elements. For instance, the 3D 20x20x2.5 mesh shows a nearly flat line, indicating minimal pressure variation along the contact region, which could be beneficial for applications requiring steady and predictable contact behaviour.

The 3D 5x5x2.5 mesh shows higher pressures values compared to the 10x10x2.5 mesh, which suggests that even within 3D modeling, smaller mesh sizes might be more responsive to nuances in contact mechanics.

The diagrams in Figures 14 and 15 illustrate that using the smallest mesh size in finite element analysis introduces computational instability, as evidenced by the significant fluctuations in both stress and contact pressure. These fluctuations indicate that the finer mesh size, while potentially capturing more detailed phenomena, can lead to unstable numerical results.

4. CONCLUSION

This study systematically investigated the impact of friction coefficient on stress distribution and contact pressure across different mesh sizes and element types in finite element simulations. The findings indicate that the friction coefficient significantly influences both contact pressure and stress distribution, with higher coefficients promoting a more even distribution of load across the contact surfaces. This trend was consistent regardless of the mesh size or element type, although finer meshes provided more detailed insights into localized behaviours.

Main conclusions from this study are:

- Increased friction generally reduces peak stresses and pressures, potentially reducing wear and fatigue in mechanical components.
- Finer meshes capture more detailed phenomena, essential for applications requiring high precision, whereas coarser meshes are sufficient for broader, less detailed structural analyses.
- Both 3D and shell elements are effective in capturing the essential trends, but the choice between them should be based on the specific requirements of computational efficiency and detail level needed.

The insights gained underscore the crucial role of friction in the mechanical integrity of contacting surfaces, highlighting how changes in friction coefficient can significantly alter stress and pressure distributions.

The future research will include more complex geometries and sophisticated modeling techniques to further enhance understanding of material interactions in a wider array of engineering applications. This expanded geometric complexity is critical for applications in diverse fields such as automotive and aerospace industry and precision engineering, where contact surfaces are rarely perfectly flat.

The aim is to enhance the predictability and accuracy of finite element analyses under varied and realistic loading conditions, thereby providing more detailed guidance for the design and optimization of components with complex contact geometries.

ACKNOWLEDGEMENT

This research is partly supported by the Ministry of Science, Technological Development and Innovation, Republic of Serbia, Agreement No. 451-03-136/2025-03/200378, and by the Science Fund of the Republic of Serbia, #GRANT No 7475, Prediction of damage evolution in engineering structures - PROMINENT.

ORCID iDs

Vladimir MILOVANOVIC 0000-0003-3071-4728
Milos PESIC 0000-0002-3405-5216
Snezana VULOVIC 0000-0001-5784-0906
Zivana JOVANOVIC PESIC 0000-0002-1373-0040
Miroslav ZIVKOVIC 0000-0002-0752-6289

REFERENCES

[1] J. C. Simo and T. J. R. Hughes “Computational Inelasticity,” *Springer,* 2006.
[2] H. J. Sutherland, Y. Zhang and S. Song “Finite element simulation of frictional contact problems: A review,” *International Journal for Numerical Methods in Engineering,* vol. 107, no 11, pp. 907-937, 2016.
[3] O. C. Zienkiewicz and R. L. Taylor, “The Finite Element Method (6th ed.),” *Butterworth-Heinemann*, 2005.
[4] T. J. R. Hughes, “The Finite Element Method: Linear Static and Dynamic Finite Element Analysis,” *Dover Publications,* 2012.
[5] D. Oliveiro, F. Gagliardi and M. Di Natale, “Friction and wear in mechanical contacts: numerical modeling and experimental validation,” *Tribology International, 151, 106516,* 2020.
[6] Z. Li, X. Li, L. Zhang “Impact of friction coefficient on the behavior of contact problems in finite element analysis: A numerical study,” *Journal of Computational and Applied Mathematics,* vol. 338, pp. 147-158, 2018.
[7] Femap, 2021. Finite Element Modelling and Post-Processing Application FEMAP v2021.2, Siemens.
[8] Pijpers, R.J.M., and Slot, H.M. “Friction coefficients for steel to steel contact surfaces in air and seawater,” *SINTEF Publications*, 2021.