IETI Transactions on Engineering Research and Practice

http://ieti.net/TERP/

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

A COMPARATIVE STUDY OF LINEAR CONTACT PROBLEMS IN SOFTWARE SIMCENTER FEMAP WITH NASTRAN

Miloš Pešić[1, a], Vladimir Milovanović[1,b], Lidija Jelić[1,c] and Nikola Jović[1,d]

[1,2,3,4]University of Kragujevac, Faculty of Engineering, Kragujevac, Republic of Serbia

[a]milospesic1736@gmail.com, [b]vladicka@kg.ac.rs, [c]jjelic.95@gmail.com, [d]njovic1995@gmail.com

Abstract The aim of this article is comparative analysis of results obtained using the linear contact between the two plates set at an angle of 90° in software Simcenter Femap with Nastran. For consideration of setted problem two models of different dimensions and load values, with different type of elements and FEM mesh density were analyzed. The boundary conditions and the mode of action of the load are the same in all considered FEM models. For comparative analysis in FEM models different combinations of 2D-shell elements of the appropriate thickness and 3D elements (hexahedral) were used for analysis. The changes in stress values and node displacements in the contact regions in dependence on the FEM mesh density and different type of elements in contact regions were considered.

Keywords: Linear contact analysis; finite element method; stress contact distribution.

1. INTRODUCTION

Contact mechanics is part of solid mechanics, and the main study of it is the deformation of solids that touch each other at one or more points. The general distinction in contact mechanics is between stresses that act perpendicular to the contacting bodies' surfaces and frictional stresses that act tangentially between the surfaces.

The determination of stresses and deformations as a consequence of the solid bodies interaction has important role in many engineering problems. The understanding and correctly solving contact interactions is crucial for the safe design and safe life of different types of engineering structures. There are many analytical solutions for solving contact problems but only for bodies with simple geometries and loading conditions. However, in practice the contact problems are highly complex and some types of extension of the analytical solutions for solving contact problems are difficult and unreliable. Because of that numerical solution techniques for contact problems have evolved significantly over the last decades and can solve complex engineering problems.

The Finite element method (FEM), as numerical method, is commonly used for solving contact problems in solid mechanics [1]. FEM analysis has also a wide application in biomechanics and numerical simulation of the contact between the sensor head and the soft tissue, and that had been analyzed in [2]. Some authors confirm that in the railway industry, when analyzing the contact between the Rail-Wheel, using 3D elements give a good match with the real conditions of exploitation [3]. Numerical examples to demonstrate a comparison of the presented algorithms when applied to contact problems are shown in [4]. The authors in [5] confirm that multi-level multi-summation (MLMS) is more advantageous than the fast Fourier transform (FFT) for solving three-dimensional concentrated contact problems.

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

For solving all these contact problems authors in their studies used different types of contacts and different combinations of FEM elements with different FEM mesh density. Because of that, this paper has for aim to present the comparative study of obtained results for a linear contact test in Simcenter Femap with Nastran software [6], between the two plates set at an angle of 90° for two models with different dimensions, with different type of elements and FEM mesh density.

2. CONTACT TYPES AND TEST FEM MODELS

In contact problems there are three types of contact: surface, line, and point contact. In this paper, contact between two bodies on the surface was examined. The main goal of this paper is to monitor stress change and node displacements in contact regions depending on the FEM mesh density and different combinations of 2D-shell elements of the appropriate thickness and 3D elements (hexahedral) for modelling bodies in contact.

For this study models with two geometries were tested. The values of the forces are different, but the constraints and the way the load acts are the same. All parameters to define contact property are setted to default, except friction coefficient which is 0.5 according to [7] because steel to steel dry contact were considered. For all FEM models for testing and comparative study linear static analysis was performed. Both plates are made from steel and have the same material characteristics (2.1 105 MPa as Young Modulus, 7.85 106 kg/mm3 as density and 0.3 as Poisson ratio).

As already mentioned, models with two geometries have been prepared to monitor stress change in contact regions using different types of finite elements and different FEM mesh density.

The dimensions of the plate panels are 120x60x5 mm for the first geometry model. The total load force is 50 kN and it is acting in the negative vertical direction, whereas the load itself is given as the pressure on the elements surface. For this geometry model it is considered twelve FEM models with different FEM mesh density and different combinations of elements for modelling bodies in contact. All Test FEM models for this type geometry are shown in Table 1 and all three dimensions (length, width and thickness) of finite element are approximately equal.

Table 1. Test FEM models for first geometry model (plate panels 120x60x5 mm).

Finite element type combination	Finite elements size	Mark of FEM model
3D elements - Bottom plate & 3D elements - Top plate	10x10x5 mm	3D-3D-M1-1
	5x5x5 mm	3D-3D-M1-2
	2.5x2.5x5 mm	3D-3D-M1-3
2D-shell elements - Bottom plate & 2D-shell elements - Top plate	10x10x5 mm	2D-2D-M1-4
	5x5x5 mm	2D-2D-M1-5
	2.5x2.5x5 mm	2D-2D-M1-6
3D elements - Bottom plate & 2D-shell elements - Top plate	10x10x5 mm	3D-2D-M1-7
	5x5x5 mm	3D-2D-M1-8
	2.5x2.5x5 mm	3D-2D-M1-9
2D-shell elements - Bottom plate & 3D elements - Top plate	10x10x5 mm	2D-3D-M1-10
	5x5x5 mm	2D-3D-M1-11
	2.5x2.5x5 mm	2D-3D-M1-12

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

One of Test FEM models for the first considered geometry when both plate panels (bodies in contact) are modelled with 3D elements (hexahedral eight nodes) size 10x10x5 mm with corresponding boundary conditions (both plate panels are clamped) and loads is shown in Figure 1.

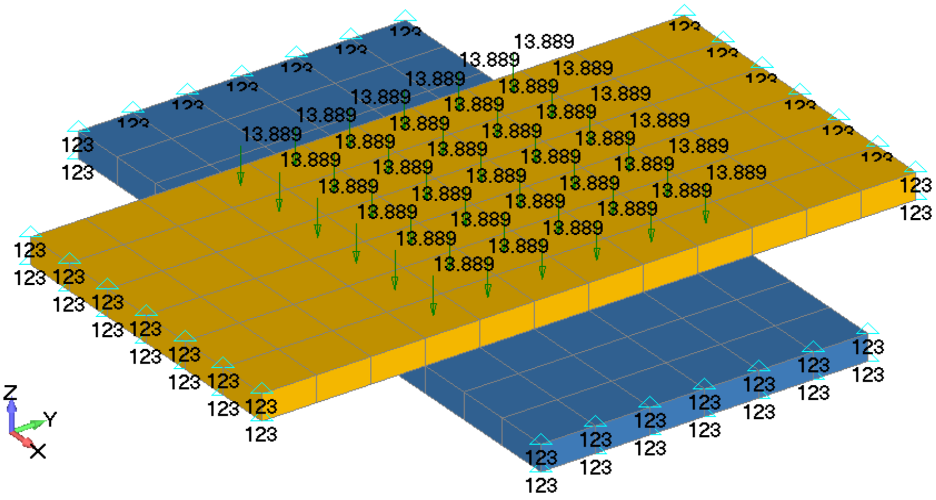

Figure 1. Geometry Model 1 - 3D-3D-M1-1 with given loads and constraints.

The dimensions of the plate panels are 640x320x5 mm for the second geometry model. The total load force is 150 kN and it is acting in the negative vertical direction, whereas the load itself is given as the pressure on the elements surface. For this geometry model it is considered twelve FEM models with different FEM mesh density and different combinations of elements for modelling bodies in contact. All Test FEM models for this type geometry are shown in Table 2 and two dimensions of finite elements (length and width) are significantly greater then thickness. This was done in order to consider some type of thin-walled structure. For this type of FEM analysis usually 2D-shell elements are required.

Table 2. Test FEM models for second geometry model (plate panels 640x320x5 mm).

Finite element type combination	Finite elements size	Mark of FEM model
3D elements - Bottom plate & 3D elements - Top plate	40x40x5 mm	3D-3D-M2-13
	20x20x5 mm	3D-3D-M2-14
	10x10x5 mm	3D-3D-M2-15
2D-shell elements - Bottom plate & 2D-shell elements - Top plate	40x40x5 mm	2D-2D-M2-16
	20x20x5 mm	2D-2D-M2-17
	10x10x5 mm	2D-2D-M2-18
3D elements - Bottom plate & 2D-shell elements - Top plate	40x40x5 mm	3D-2D-M2-19
	20x20x5 mm	3D-2D-M2-20
	10x10x5 mm	3D-2D-M2-21
2D-shell elements - Bottom plate & 3D elements - Top plate	40x40x5 mm	2D-3D-M2-22
	20x20x5 mm	2D-3D-M2-23
	10x10x5 mm	2D-3D-M2-24

IETI Transactions on Engineering Research and Practice

http://ieti.net/TERP/

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

One of Test FEM models for the second considered geometry when both plate panels (bodies in contact) are modelled with 3D elements (hexahedral eight nodes) size 40x40x5 mm with corresponding boundary conditions (both plate panels are clamped) and loads is shown in Figure 2.

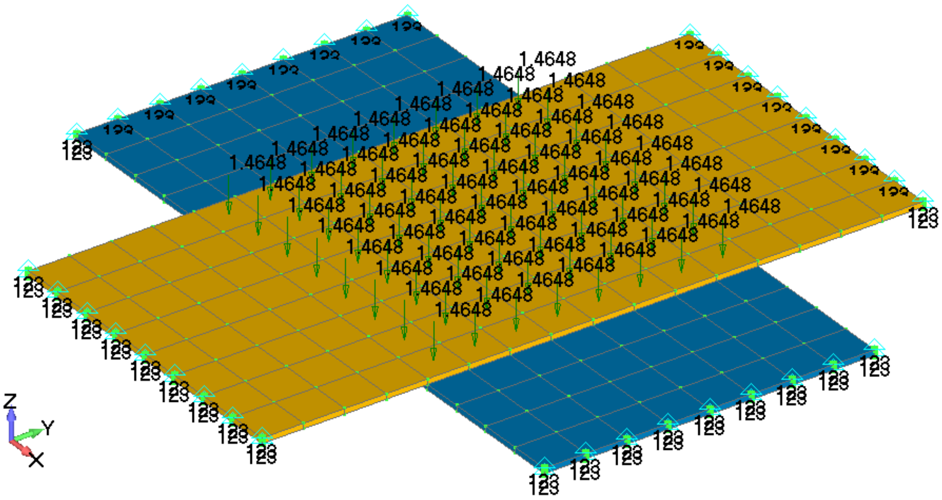

Figure 2. Geometry Model 2 - 3D-3D-M2-13 with given loads and constraints.

3. RESULTS AND DISCUSIONS

According to tasks setted in chapter 2 in order to monitor stress change and node displacements in contact regions depending on the FEM mesh density and different combinations elements all Test FEM models were analyzed. The results at the contact region showed unrealistic – very high values of stress in 2D-shell elements in both geometry models and in all mesh density combinations. For further analysis, only the contact stress values obtained in contact region, where 3D elements used for modeling bottom plate panels were taken for comparative study.

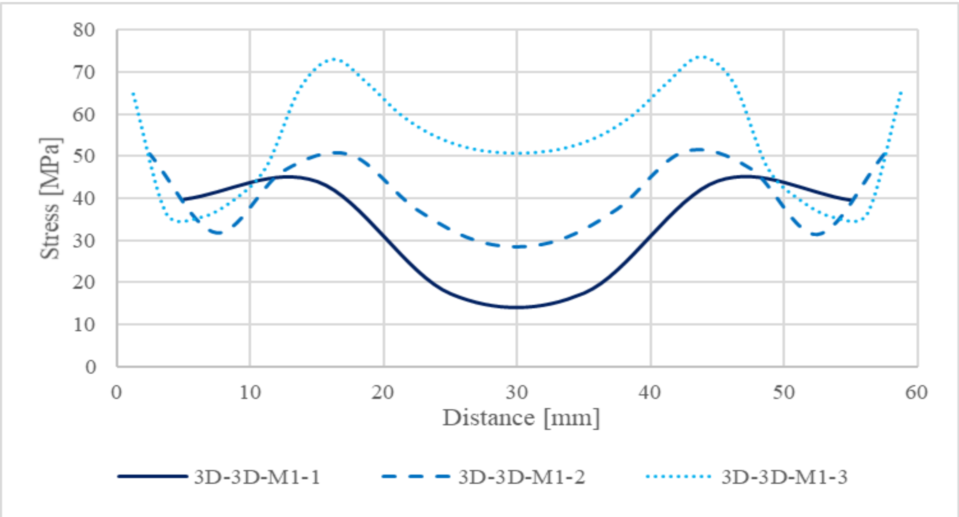

Figure 3. Stress distribution in contact region in dependence of FEM mesh density – Geometry model 1.

IETI Transactions on Engineering Research and Practice

http://ieti.net/TERP/

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

Figure 3 shows the stress distribution in the contact region in bottom plate panel, when both plate panels are modelled with 3D elements and with different FEM mesh density for geometry model 1. From the stress distributions along contact region shown in Figure 3 it is clear that with increasing FEM mesh density, stresses in considered contact region have higher values. The same trend occurs when top plate panel is modelled with 2D-shell elements. A comparative review of the obtained stresses in the contact region of the bottom plate panel when the load is transferred using 3D elements or 2D-shell elements is shown in Figure 4.

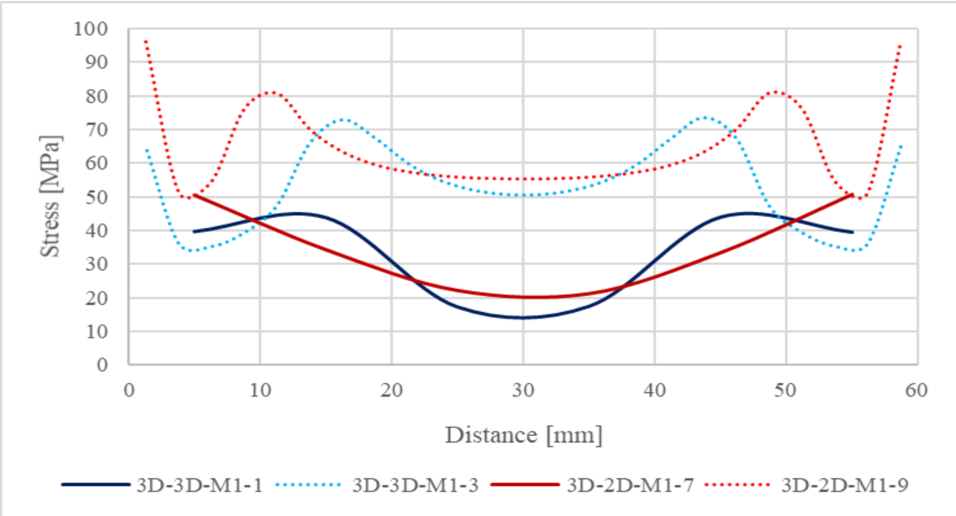

Figure 4. Stress distribution in contact region in dependence of using FEM elements for modelling contact regions - Geometry model 1.

As can be seen from the diagram in Figure 4, the values of stress oscillations in the contact regions are smaller with the combination of 3D elements for contact pairs.

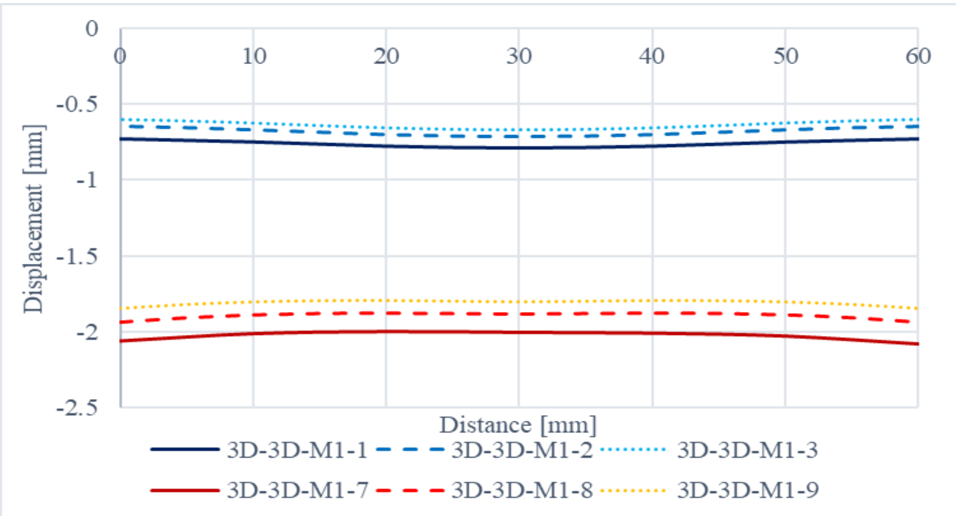

Figure 5. Nodal displacement distribution in contact region in dependence of using FEM elements for modelling contact regions - Geometry model 1.

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

A comparative review of the nodal displacements in the contact region of the bottom plate panel, for Test FEM models when both plate panels are modelled with 3D elements and with different FEM mesh density and when top plate panel is modelled with 2D-shell elements is shown in Figure 5. The results shown in Figure 5 display that the smallest nodal displacements in the contact region of the two plate panels are in case when 3D elements are used for modelling both plate panels with the smallest finite element dimensions. Higher values of the nodal displacements in the contact region are when the load is transferred from 2D-shell elements to 3D elements.

Figure 6 shows the stress distribution in the contact region in bottom plate panel, when top plate panel is modelled with 2D-shell elements while bottom plate panel is modelled with 3D elements and with different FEM mesh density for geometry model 2.

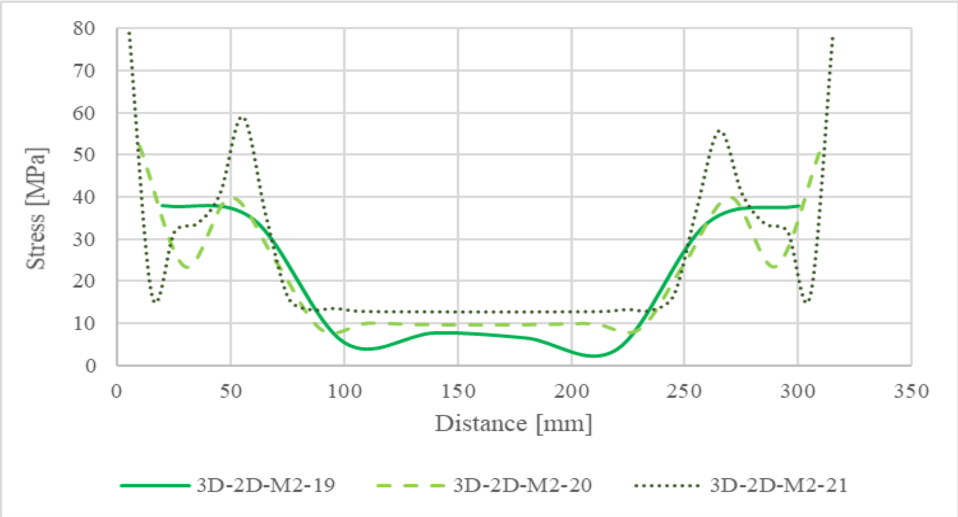

Figure 6. Stress distribution in contact region in dependence of FEM mesh density – Geometry model 2.

From the stress distributions along contact region shown in Figure 6 it is clear that with increasing FEM mesh density, stresses in considered contact region have higher values. The same trend occurs when both plate panels are modelled with 3D elements.

http://ieti.net/TERP/

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

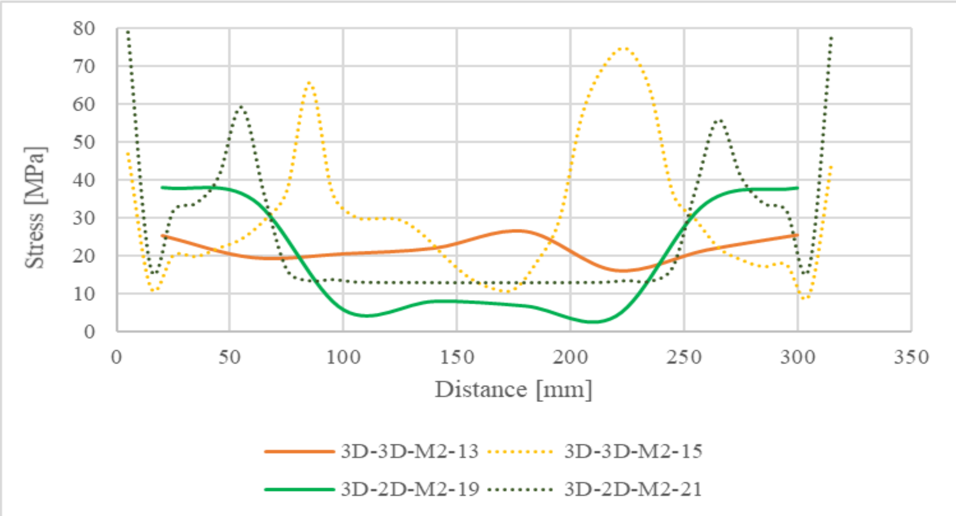

Figure 7. Stress distribution in contact region in dependence of using FEM elements for modelling contact regions - Geometry model 2.

A comparative review of the obtained stresses in the contact region of the bottom plate panel when the load is transferred using 3D elements or 2D-shell elements is shown in Figure 7. From results shown in Figure 7 it can be noticed that in Test FEM models when both plate panels are modelled with 3D elements there is an asymmetry in obtained stress results in contact region. With increasing FEM mesh density for Test FEM models when both plate panels are modelled with 3D elements it can be noticed larger asymmetry.

A comparative review of the nodal displacements in the contact region of the bottom plate panel, for Test FEM models when both plate panels are modelled with 3D elements and with different FEM mesh density and when top plate panel is modelled with 2D-shell elements is shown in Figure 8.

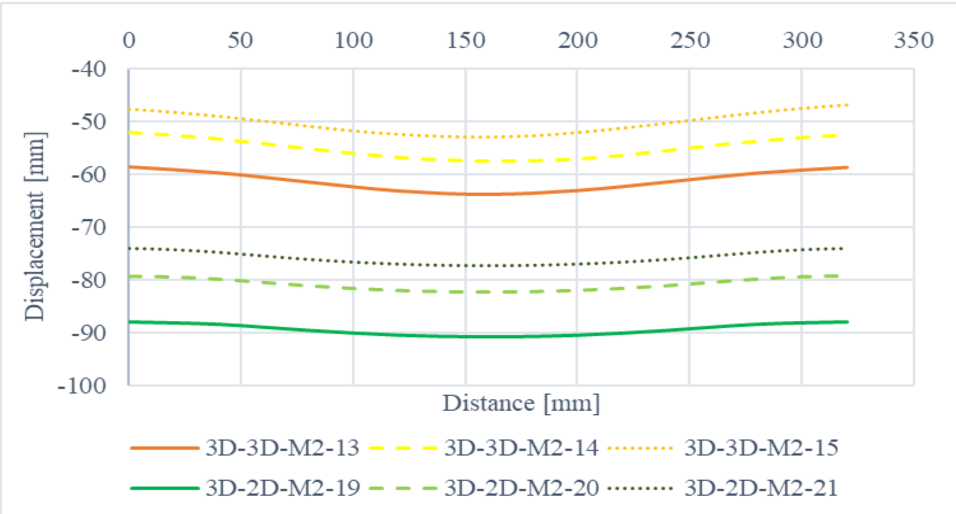

Figure 8. Nodal displacement distribution in contact region in dependence of using FEM elements for modelling contact regions - Geometry model 2.

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

The results shown in Figure 5 display that the smallest nodal displacements in the contact region of the two plate panels are in case when 3D elements are used for modelling both plate panels with the smallest finite element dimensions. Higher values of the nodal displacements in the contact region are when the load is transferred from 2D-shell elements to 3D elements.

4. CONCLUSION

Based on comparative analysis of results obtained using the linear contact between the two plates in software Simcenter Femap with Nastran with different type of elements and FEM mesh density the following conclusions were deduced:

• with increasing of FEM mesh density the contact stress values in contact regions, for all cases were higher, but the nodal displacement values in contact regions, for all cases are lower;

• comparing 2D-shell elements and 3D elements for modelling plate panel for load transfer lower stress values in contact region were obtained when both plate panels are modelled with 3D elements;

• For the first geometry model, similar results were obtained when both plate panels were modelled with 3D elements, but for the second geometry model, similar results were obtained with a combination of 3D elements on bottom plate panel and 2D-shell elements on top plate panel.

• When both plate panels were modelled with 3D elements for the second geometry model, there is an asymmetry in the stress distribution in the contact region.

This comparative study represents main base for future research in the field of contact problems solving for finding a solution that gives similar results when contact regions for both plate panels are modelled with 2D-shell elements and 3D elements.

Acknowledgements

The authors gratefully acknowledge partial support by Ministry of Education, Science and Technological Development, Republic of Serbia, Grant TR32036.

References

[1] Chouly F., and Hild P., 2013, A nitsche-based method for unilateral contact problems: Numerical analysis, *SIAM Journal on Numerical Analysis*, 51(2), p.p. 1295-1307.
[2] Dosaev M., Samsonov V., and Bekmemetev V., 2020, Comparison between 2D and 3D Simulation of Contact of Two Deformable Axisymmetric Bodies, *International Journal of Nonlinear Sciences and Numerical Simulation*, 21(2), p.p. 123-133.
[3] Arslan M.A., Kayabaşi O., 2012, 3-D Rail-Wheel contact analysis using FEA, *Advances in Engineering Software*, 45(1), p.p. 325-331.
[4] Vulovic S., Zivkovic M., Grujovic N., and Slavkovic R., 2007, A comparative Study of Contact Problems Solution Based on the Penalty and Lagrange Multiplier Approaches, *Journal of the Serbian Society for Computational Mechanics*, 1(1), p.p. 174-183.

2021, Volume 5, Issue 1, 46-54, DOI 10.6723/TERP.202102_5(1).0006

[5] Polonsky I.A., Keer L.M., 2000, Fast methods for solving rough contact problems: A comparative study, *Journal of Tribology*, 122(1), p.p. 36-41.
[6] Femap, 2019, Finite Element Modeling and PostProcessing Application, FEMAP v12, Siemens.
[7] Engineering ToolBox, *Friction and Friction Coefficients*, 2004, Available at: https://www.engineeringtoolbox.com/friction-coefficients-d_778.html