WELD DEFECT ANALYSIS IN A PRESSURE PIPELINES USING 2D AXISYMMETRIC FINITE ELEMENT ANALYSIS IN ABAQUS

UDC:621.643.411:519.6
Original scientific paper https://doi.org/10.46793/adeletters.2025.4.1.5

Muhammad Sohail Ashraf[1*]

[1]University of Engineering and Technology, Department of Mechanical Engineering, Lahore, Pakistan

Abstract:
Weld defects challenge the structural integrity of internally loaded pipelines used for pressurized oil and gas transportation. The butt weld was examined in ABAQUS software considering the 2D axisymmetric finite element analysis (FEA) modeling technique assuming identical behavior along the circumferential direction. The principal objectives were to investigate the stress concentration behavior of butt weld, to conduct mesh convergence analysis to ensure FEA accuracy, such as optimal mesh element type and size, to analyze the stress concentration effect with weld defect depth, and to determine the stress distribution along the defect edge. The methodology included modeling the pipeline system in three components: left pipe, butt-weld bead, and right pipe. The mesh convergence analysis and parametric analysis on weld geometry showed that quadrilateral quadratic elements with a 0.1 mm mesh size provide stabilized results at around 232.637 MPa with minimal variation. Parametric analysis depicts the direct correlation between stress concentration and weld defect depth. It showed that at deeper defect depths, higher stress concentrations can cause potential failure and crack initiation. This study concludes the significance of carefully selecting mesh element types and sizes. Quadrilateral quadratic elements capture detailed stress distribution in the curved geometries. The stress produced in weld defects is lower than the material's strengths and is within safe limits. This research highlights the importance of managing the integrity and safety assessment of pipelines.

ARTICLE HISTORY
Received: 2 February 2025
Revised: 18 March 2025
Accepted: 27 March 2025
Published: 31 March 2025

KEYWORDS
Weld defect, Pipeline, 2D Axisymmetric, Finite Element Analysis, ABAQUS

1. INTRODUCTION

Pressurized pipes are widely used in nuclear reactors, chemical industries, and marines, subjected to extreme mechanical loads, and therefore, advanced methods are used for their analysis [1]. Pressurized gases, such as compressed natural gas and liquids, are transported through pipelines and exposed to the natural environment. Due to drastic surrounding conditions, degradation in pipes can cause accidents such as explosions and leakage [2]. In repairing and joining pipes, butt joint welding is commonly used due to several reasons, such as economical, space-efficient, providing better sealing than bolt joints, etc. [3]. Increasing the working pressure in pipelines enhances the oil and gas transportation efficiency but makes the weld defects between the pipelines more prominent [4]. One of the most common problems associated with butt joint welding is incomplete penetration of filler metal. This is caused by insufficient input heat to melt base metal for fusion, incorrect torch angle, electrode manipulation, improper welding technique, thick root face, and small root gap, which introduce the defect. These defects serve as stress concentration points for crack propagation. Ultimately, weld defects reduce the safety of pipelines and increase the risk of

*CONTACT: Muhammad Sohail Ashraf, e-mail: sohailashraf858@gmail.com

failure, which may cause damage to the surroundings [5]. Therefore, it is necessary to study the safety and structural integrity of the butt joint weld.

Finite Element Analysis (FEA) is a numerical computational technique that approximates the behaviour of a structure by dividing it into discrete elements [6, 7]. The 2D axisymmetric FEA technique is advantageous among the various modeling techniques in simulating cylindrical structures such as pipelines, pressure vessels, nozzles, etc., which have rotational symmetry along a central axis [8]. A 2D cross-section is modeled instead of a complete 3D model, reducing computational complexity while maintaining accuracy [9]. Assumption of identical behavior along the circumferential direction is made in the analysis to efficiently predict stress, deformation, and strain using a powerful tool such as ABAQUS. Meshing is used in FEA to divide the structure into small elements and solve them individually. Accurate structure meshing depends on the selection of element shapes and types. Analyzing the incomplete penetration weld defect in pressure pipelines using an axisymmetric finite element modeling technique under various loading conditions is a novel approach [10].

One of the key advantages of FEA over traditional methods is its ability to account for complex boundary conditions and material heterogeneities in a more realistic manner [11, 12]. Traditional analytical methods, such as those based on the ASME Boiler and Pressure Vessel Code or WRC 297, often rely on simplifications that may not fully capture the stress distributions in complex weld geometries or areas with material defects [13]. In contrast, FEA models can incorporate real-world geometries and boundary conditions (e.g., internal pressure, thermal stresses, and weld residual stresses), offering more accurate predictions of a pipeline's response to loading conditions. This makes FEA an indispensable tool for evaluating the safety and reliability of pipelines subjected to high internal pressures and harsh environmental conditions [14]. Additionally, the mesh convergence study is an essential aspect of FEA, ensuring that the numerical results are accurate and stable as the mesh is refined. Mesh convergence studies help to identify the optimal mesh density required to obtain reliable results without unnecessarily increasing computational cost [15]. Without conducting a proper convergence study, the FEA results may be subject to numerical errors, leading to inaccurate stress predictions [16]. This is particularly important when modeling weld defects, where stress concentrations can vary significantly over small distances, requiring fine meshing in critical areas to capture the stress gradients accurately.

Various authors have underscored the importance of accurately assessing the structural integrity of pipelines with welded joints subjected to operational stresses. Čamagić et al. [17] conducted comprehensive studies on welded joints, highlighting how temperature and operational periods significantly affect material brittleness, fracture resistance, and structural integrity under impact loading conditions, crucially affecting the durability and maintenance of pressurized components. Nikolić et al. applied finite element analyses to evaluate stress concentrations in steel plates with geometric discontinuities, such as rectangular openings, concluding that stress concentrations significantly depend on the orientation and geometry of openings relative to applied loads [18]. Additionally, critical aspects such as geometry and position of discontinuities, like weld defects or holes, have profound implications for stress distribution and concentration, necessitating meticulous design considerations in pipeline systems [19]. Compliance with international standards such as ASTM E23-02, SRPS EN ISO 9692-1:2012, and SRPS EN ISO 9692-2:2008 provides rigorous guidelines for weld preparation, evaluation methods, and the necessary procedures to maintain structural reliability under various loading conditions, thus ensuring consistency and safety in pipeline designs and operations [20].

Lu et al. [5] provided a novel model of incomplete penetration defect in a girth weld and derived a residual strength equation based on the analytical solution obtained from the stress function method. Analytical results obtained were verified by the Finite Element Method (FEM) and experimental results [5]. Dai et al. [21] studied the structural integrity of repaired weldments in a pressurized pipeline. Residual stresses in a 3D model having a Butt joint were investigated by thermal elastic-plastic FEM [21]. Teran et al. [22] conducted a Finite Element Analysis (FEA) to predict the failure pressure for corrosion defects in the base pipe and weld bead. Developed a model with rectangular defect and conducted FEA under various changes in length (L), depth (D), and width (W). They concluded that failure pressure decreases with an increase in L, D, and W of corrosion defect [22]. Rybicki et al. [23] conducted a comprehensive FE analysis in ADINA to determine the residual and hoop stresses in single-pass butt-welded pipe. The study focused on the effect of the weld cooling rate on thermal stresses, circumferential stresses, and axial stresses without an axisymmetric assumption approach [23]. Heckman [6] focused on the FEA of pressure vessels to evaluate their structural integrity under various loading conditions using numerical simulations. Their study provides insights into the challenges of modeling welded structures, particularly those subjected to internal pressure. It highlights the critical areas where stress concentration occurs, such as at weld joints and discontinuities [6]. Cui and Qie [24] investigated the inconsistent pipe thickness when two pipes are joined. The Novel Smoothed Finite Element Method (S-FEM) was used to analyze the stress of a defect under axisymmetric conditions in pressure pipelines using ABAQUS. The effect of the weld bead on defect was neglected in this study, and it concluded that for thinner sections, maximum hoop stress is only affected by thickness mismatch [24].

The present study investigates the stress distribution along the weld defect in pressure pipelines using the 2D axisymmetric FEA modeling technique. The first objective of this study is to conduct a mesh sensitivity analysis to determine the FEA accuracy in predicting the maximum Von Mises stress along the weld edge. The second objective is to study the effect of meshing element shapes and polynomial orders in predicting the maximum Von Mises stress along the weld defect edge. The third objective is to conduct a parametric analysis of a weld defect at various weld defect depths at optimum mesh element shape and polynomial order. The fourth objective of this study is to determine the Von Mises stress distribution along the weld defect edge of a butt joint subjected to intense internal pressure.

2. MATERIAL AND METHODS

The Finite Element Method (FEM) was used to investigate the behavior of weld defects in a pressurized pipeline system using a 2D axisymmetric modeling approach in ABAQUS. Fig. 1 shows that the model consists of three components: left pipe, weld bead, and right pipe. Pipes were modeled as 2D axisymmetric, deformable shells with lengths of 75 mm and 18.2 mm pipe thickness. One side of each pipe was at an angle 30^o. Similarly, the weld bead was modeled to represent a typical butt-weld joint with sides forming 60^o angle and a lower thickness of 5 mm. The defect was introduced in the lower side of the butt-weld bead.

For simplicity, the weld bead, left, and right pipe material was considered homogeneous and isotropic, with a Young's Modulus of 2.1×10^{11} N/m^2 and a Poisson's ratio of 0.3, consistent with typical steel properties used in high-pressure pipelines [24]. The left pipe-inclined edge, weld bead, and right pipe-inclined edges were tied using a constraint module to connect them seamlessly. This constraint ensures that the pipeline system behaves as a single entity.

An internal uniform pressure load of 50 MPa was applied along the internal length of the pipeline system. The boundary conditions were applied at the lower end of the pipeline system to restrict its displacement in the Y-direction and rotation along the Z-axis.

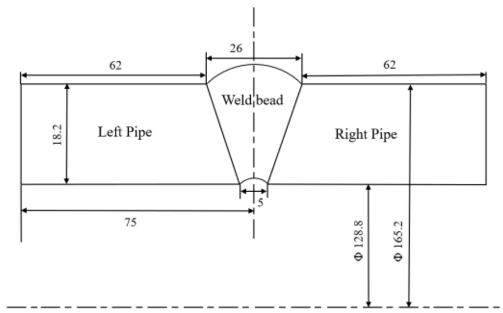

Fig. 1. Weld groove dimensions in a pressurized pipeline system

A mesh convergence analysis was conducted to not rely on a single result. Both quadrilateral and triangular element shapes were tested with linear and quadratic polynomial orders to determine maximum Von Mises results convergence with the mesh element size. For the mesh convergence study, the mesh element size was varied from 0.1 mm to 0.5 mm with a step of 0.1 mm. Results were evaluated for the optimal mesh element size, shape, and type for accurate stress distribution modeling. Results were recorded in graphical form. Fig. 2 shows the isometric view of the butt weld joint in a pressurized pipeline system.

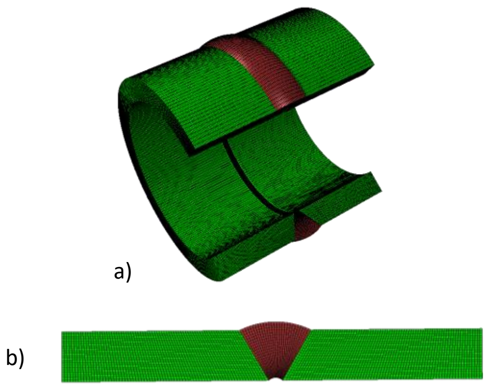

Fig. 2 Butt-weld repairing in a pressurized pipeline system

A parametric analysis study investigated the dependence of Von Mises's stress distribution on defect size (depth). A weld defect was introduced by increasing its depth from 0 to 1 mm with a jump of 0.25 mm. Corresponding maximum stresses to the weld depth were recorded in graphical form.

2.1. Model Validation

The 2D axisymmetric butt-weld joint model shown in Fig. 1 was validated by ensuring the accuracy of its geometric framework, which was adopted from research [21]. The study employs a different numerical approach, material properties, and analysis objectives, while the model geometry was maintained consistently to ensure realistic defect representation. The numerical validation process includes a mesh convergence study, confirming that 0.1 mm quadrilateral quadratic elements provide stable stress values. Additionally, boundary conditions were verified from research [24], ensuring that the applied constraints and loading conditions are real-world operating conditions.

3. RESULTS AND DISCUSSION

This study conducted a comprehensive analysis of the butt-weld defect using the 2D axisymmetric finite element analysis modeling technique in ABAQUS.

3.1 Mesh Convergence Analysis

Maximum Von Mises stress was predicted along the edge of the weld defect in a pipeline system. The mesh convergence analysis was conducted using quadrilateral and triangular elements with a polynomial order of linear and quadratic by varying mesh element size from 0.1 mm to 0.5 mm with an increment of 0.1 mm, as shown in Figs. 3 and 4. The results revealed that the quadrilateral quadratic mesh element type provides the stabilised maximum Von Mises stress results and indicates convergence. The results show that the quadrilateral square gave accurate stress distribution results along the seam defect's edge.

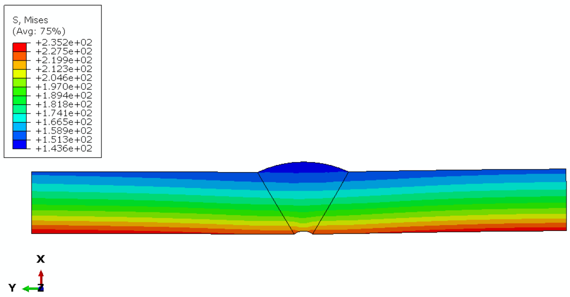

Fig. 3. Von Mises Stress at 0.5 mm mesh element size

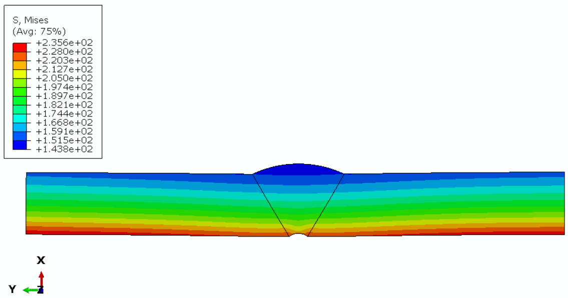

Fig. 4 Von Mises Stress at 0.1 mm mesh element size

Fig. 5 shows that at a 0.2 mm mesh size, quadrilateral quadratic elements provide a stabilized value of Von Mises stress of 232.71 MPa, and minimal deviation was observed when the mesh size was further reduced. However, 0.1 mm mesh size was optimal following computational efficiency. At the same time, triangular mesh element shapes, specifically with linear polynomial order, require a significant reduction in mesh size for convergence. Polynomial order plays a significant role in mesh convergence analysis. It is observed that quadratic mesh element size performs faster convergence and provides accurate results. Meanwhile, linear mesh element types slowly capture stress gradients. Quadrilateral quadratic elements show consistently outstanding performance at all mesh sizes compared to triangular element shapes.

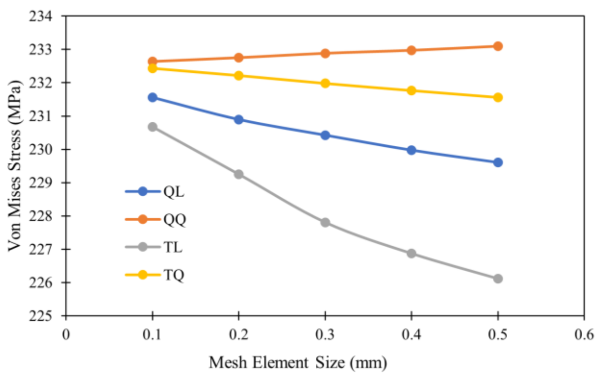

Fig. 5 Mesh convergence analysis at different mesh element types and sizes

3.2 Parametric Analysis: Effect of Weld Defect Depth

Imperfections in the welding processes, environmental factors, and small root gaps introduce defects in the weld bead, such as incomplete penetration of weld material in the gap. This incomplete penetration defect is shown in Fig. 1. A parametric study was conducted to investigate the effect of weld defect depth on the maximum Von Mises stress. Results demonstrated a direct relationship between maximum Von Mises stress and butt-weld defect depth.

Maximum Von Mises stress increases with the increase in depth from 226.039 MPa at 0 mm depth to 232.637 MPa at 1 mm depth from Fig. 8, and stress contours are shown in Figs. 6 and 7, respectively. The increasing stress trend indicates a high risk of stress concentration and potential failure at deeper weld defects, as shown in Fig. 8. The parametric study highlights the high risk of stress concentration and potential failure at more profound weld defects. Since the Von Mises stress at 1 mm depth is less than the material's.

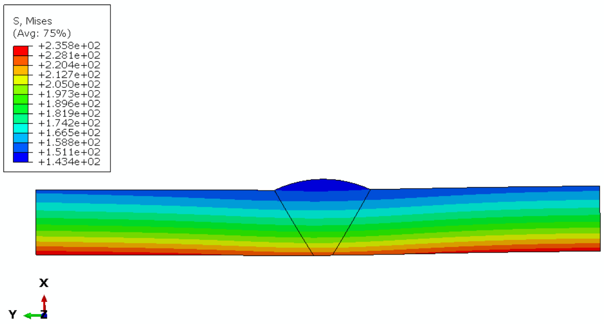

Fig. 6. Butt-weld with no defect in a pressurized pipeline system

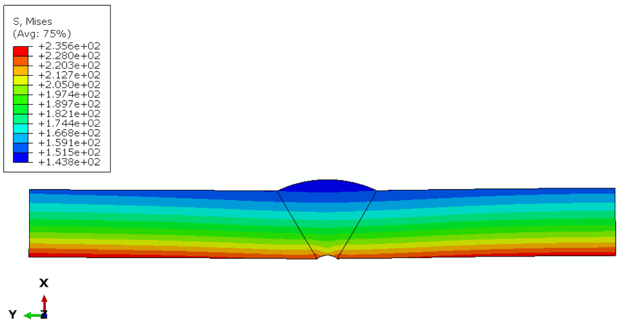

Fig. 7. Butt-weld with 1 mm defect in a pressurized pipeline system

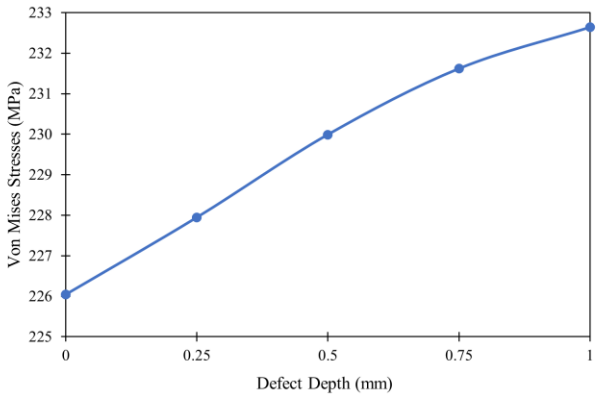

Fig. 8. Parametric Analysis of a weld defect

Fig. 9 provides the Von Mises stress distribution along the edge of the butt-weld under the various defect depths in a pressurized pipeline system. Maximum stress occurs at sharp edges formed due to imperfections in the weld bead. This leads to a geometric discontinuity, creating localized stress concentrations under the applied internal pressure. This stress distribution behavior is assumed to be the same along its axis of symmetry. The horizontal stress distribution is obtained at no defect, and higher stresses are introduced at more profound defects.

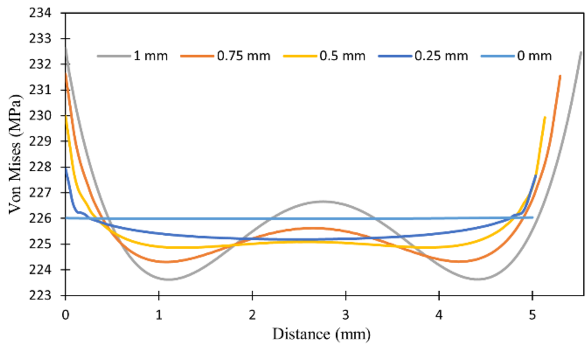

Fig. 9. Stress Variation along the Weld Defect at different depths

WRC 297 (Welding Research Council) provides stress concentration factors ranging from 1.2 to 1.8 for butt-welds [25], while the FEA results showed a maximum Von Mises stress of 232.637 MPa at the weld defect, highlighting the higher accuracy of FEA in capturing complex geometries and defect behavior.

This study plays an important role in pipeline integrity assessment, such as preventive maintenance and inspection, weld quality control, structural design optimization, failure prediction, and risk management.

4. CONCLUSION

Weld defect behavior in the pressurized pipeline system was investigated using a 2D Axisymmetric FEA modeling technique in ABAQUS. A mesh convergence analysis was conducted to determine the optimal mesh element type and size for the analysis. The quadrilateral quadratic mesh element type provides stable max Von Mises stress results at around 232.637 MPa at 0.1 mm mesh size. It shows consistent behavior as the mesh element size was decreased from 0.5 mm to 0.1 mm. Triangular element types require further refinement in mesh size for convergence, which makes them computationally expensive. At optimal mesh element type and size, parametric analysis was conducted to investigate the stress variation with weld defect depths. A parametric study reveals that at higher weld depths, stress concentration and potential failure in pressurized pipeline systems increase at deep weld defect depths. Stress distribution along the defect edge shows that maximum stress concentration occurs at corners due to abrupt geometric changes in a pipeline system. These results enhanced pipeline integrity, improved weld quality control, and optimized structural design. Results are highly applicable in preventive maintenance, failure prediction, and risk management, significantly enhancing pipeline safety and reliability. Industrial applications include pipeline safety and maintenance in the oil and gas industry, preventing failures in hazardous material pipelines in the chemical and petrochemical sectors, and strengthening pressurized pipeline systems in nuclear power plants. This research ultimately helps extend pipeline service life and ensure safer operational conditions across various industries.

The current study can be extended by employing 3D FEA instead of 2D axisymmetric modeling techniques to obtain a more accurate representation of stress distribution in the weld defect. Including anisotropic material modeling would help understand how directional mechanical properties influence stress concentration and defect propagation. Incorporating fracture mechanics-based analysis for crack propagation would improve the defect assessment and failure prediction methodologies. These advancements would enhance the reliability of numerical methods in pipeline integrity management and safety evaluation.

Conflicts of Interest

The author declares no conflict of interest.

REFERENCES

[1] X. Chen, X. Chen, D. Yu, B. Gao, Recent progresses in experimental investigation and finite element analysis of ratcheting in pressurized piping. *International Journal of Pressure Vessels and Piping,* 101, 2013: 113-142. https://doi.org/10.1016/j.ijpvp.2012.10.008

[2] Y. Da, B. Wang, D. Liu, Z. Qian, An analytical approach to reconstruction of axisymmetric defects in pipelines using T (0, 1) guided waves. *Applied Mathematics and Mechanics,* 41(10), 2020: 1479-1492. https://doi.org/10.1007/s10483-020-2661-9

[3] C. Basavaraju, Simplified analysis of shrinkage in pipe to pipe butt welds. *Nuclear Engineering and Design,* 197(3), 2000: 239-247. https://doi.org/10.1016/S0029-5493(99)00302-7

[4] L. Li, L. Zhu, X. Yan, H. Jia, S. Xie, Y. Zou, The Impact of Weld Repairs on the Microstructure and Mechanical Integrity of X80 Pipelines in Oil and Gas Transmission. *Processes,* 13(2), 2025: 512. https://doi.org/10.3390/pr13020512

[5] K.-Q. Lu, J.-Q. Lin, Z.-F. Chen, W. Wang, H. Yang, Safety assessment of incomplete penetration defects at the root of girth welds in pipelines. *Ocean Engineering,* 230, 2021: 109003. https://doi.org/10.1016/j.oceaneng.2021.109003

[6] D. Heckman, Finite element analysis of pressure vessels. *Monterey Bay Aquarium Research Institute, Moss Landing, Calif,* 1998.

[7] G. R. Liu, S.S. Quek, The finite element method: a practical course. *Butterworth-Heinemann*, Oxford, United Kingdom, 2013.

[8] J. Mackerle, Finite element analysis of fastening and joining: A bibliography (1990–2002). *International Journal of Pressure Vessels and Piping,* 80(4), 2003: 253-271. https://doi.org/10.1016/S0308-0161(03)00030-9

[9] C. Lu, X. Ma, J.E. Mills, Cyclic performance of bolted cruciform and splice connectors in retrofitted transmission tower legs. *Thin-Walled Structures,* 122, 2018: 264-285. https://doi.org/10.1016/j.tws.2017.10.020

[10] J.F. Lancaster, Metallurgy of welding. *Woodhead*, 1999.

[11] C. Miehe, C. Bayreuther, On multiscale FE analyses of heterogeneous structures: from homogenization to multigrid solvers.

International Journal for Numerical Methods in Engineering, 71(10), 2007: 1135-1180. https://doi.org/10.1002/nme.1972

[12] J.D. Arregui-Mena, L. Margetts, P.M. Mummery, Practical application of the stochastic finite element method. *Archives of computational methods in engineering,* 23, 2016: 171-190. https://doi.org/10.1007/s11831-014-9139-3

[13] A.I. Eldufani, Finite element stress analysis of axisymmetric and non-axisymmetric pressure vessels and their limit loads. *The University of Manchester*, United Kingdom, 2006.

[14] M. Karimi, Finite Element Simulations of Welding Distortions and Predictions Using Deep Learning Methods. *University of British Columbia*, Vancouver, Canada, 2024.

[15] A. Ghavidel, M. Rashki, H. Ghohani Arab, M. Azhdary Moghaddam, Reliability mesh convergence analysis by introducing expanded control variates. *Frontiers of Structural and Civil Engineering,* 14, 2020: 1012-1023. https://doi.org/10.1007/s11709-020-0631-6

[16] I.-T. Wang, Numerical and experimental verification of finite element mesh convergence under explosion loading. *Journal of Vibroengineering,* 16(4), 2014: 1786-1798.

[17] I. Čamagić, S. Jović, M. Radojković, S. Sedmak, A. Sedmak, Z. Burzić, C. Delamarian, Influence of temperature and exploitation period on the behaviour of a welded joint subjected to impact loading. *Structural Integrity and Life,* 16(3), 2016: 179-185.

[18] V. Nikolić, Ć. Dolićanin, M. Radojković, Application of finite element analysis of thin steel plate with holes. *Tehnički Vjesnik,* 18(1), 2011: 57-62.

[19] S. Sanati, S.F. Nabavi, R. Esmaili, A. Farshidianfar, Laser wobble welding process: a review on metallurgical, mechanical, and geometrical characteristics and defects. *Lasers in Manufacturing and Materials Processing,* 11(3), 2024: 743-780. https://doi.org/10.1007/s40516-024-00252-x

[20] ISO 3690:2018, Welding and allied processes—Determination of hydrogen content in arc weld metal, 2018.

[21] P. Dai, Y. Wang, S. Li, S. Lu, G. Feng, D. Deng, FEM analysis of residual stress induced by repair welding in SUS304 stainless steel pipe butt-welded joint. *Journal of Manufacturing Processes,* 58, 2020: 975-983. https://doi.org/10.1016/j.jmapro.2020.09.006

[22] G. Terán, S. Capula-Colindres, J. Velázquez, D. Angeles-Herrera, E. Torres-Santillán, On the influence of the corrosion defect size in the welding bead, heat-affected zone, and base metal in pipeline failure pressure estimation: A finite element analysis study. *Journal of Pressure Vessel Technology,* 141(3), 2019: 031001. https://doi.org/10.1115/1.4042908

[23] E.F. Rybicki, D.W. Schmueser, R.W. Stonesifer, J.J. Groom, H.W. Mishler, A finite-element model for residual stresses and deflections in girth-butt welded pipes. *Journal of Pressure Vessel Technology*, 100(3), 1978: 256-262. https://doi.org/10.1115/1.3454464

[24] X. Cui Y. Qie, Using axisymmetric smoothed finite element method (S-FEM) to analyze pressure piping with defect in ABAQUS. *International Journal of Computational Methods,* 17(08), 2020: 1930001. https://doi.org/10.1142/S0219876219300010

[25] J.L. Mershon, K. Mokhtarian, G.V. Ranjan, E.C. Rodabaugh, Local Stresses In Cylindrical Shells Due To External Loadings On Nozzles-Supplement to WRC Bulletin No. 107. WRC Bulletin 297. *WRC*, New York, USA, 1987.