ISSN 1330-3651(Print), ISSN 1848-6339 (Online)
https://doi.org/10.17559/TV-20240521001636
Original scientific paper

Integrity Assessment of the Welded Nozzle-Header Joint of the Steam Superheater

Ivan DUNĐER*, Pejo KONJATIĆ, Marko KATNIĆ, Ivan GELO

Abstract: Manufacturing flaw in pressure vessels and pipelines, especially in welded joints, poses a major challenge to structural integrity and operational safety. This paper examines the effects of such flaws, focussing on a specific case study of inclusions observed at the root of welded head-nozzle joints in a steam superheater. Finite Element Analysis (FEA) is used to investigate how these flaws affect the load capacity of the pressure vessel. To simulate the behaviour of the welds under operating conditions, two scenarios are considered: one with a sharp elliptical crack between two welded nozzles and another with a full torus-shaped flaw along the entire circumference of the weld root. During the analysis, stress distributions, plastic limit loads, stress intensity factors, fracture toughness and structural integrity are assessed using the Failure Assessment Diagram (FAD) and the SINTAP method. The results show that although the presence of a sharp elliptical crack with the considered dimensions between the nozzles does not significantly affect the static strength of the structure, plastic collapse must be considered as a potential failure mechanism, especially for highly plastic materials. The FAD analysis shows that the structure can withstand the operating pressure without failing, but failure due to plastic collapse can occur at higher pressures. The study emphasises the importance of considering both fracture mechanics and the plastic failure criterion when assessing the integrity of welded components in pressure vessels.

Keywords: crack; finite element method; fracture mechanics; SINTAP procedure; the welded joint

1 INTRODUCTION

Flaws in pressure vessels and pipelines are unavoidable due to limitations in manufacturing and welding processes. Applying the principles of fracture mechanics to the design of pressure vessels and pipelines together with non-destructive testing is a common method used by engineers to solve this problem [1]. Numerous studies have been conducted on the problem of various failures. The paper [2] examined the integrity assessment of a spherical ammonia storage tank. Inspections revealed surface cracks in the welded joints. Fracture mechanics parameters were used to evaluate the tank's structural integrity. Four critical cracks were examined by calculating their stress intensity factors and comparing them to the material's fracture toughness, confirming they did not compromise the tank's safety. Further analysis using the Failure Assessment Diagram (FAD) and 3D finite element simulations validated the tank's structural reliability. The study concluded that the cracks do not threaten the tank's integrity, and the groove modifications enhanced its safety. The paper [3] was focused on assessing the integrity of welded joints in the main pipeline of a reversible hydroelectric power plant. The study combined experimental testing on a prototype, including hydrostatic and rapid fracture tests, with numerical simulations using the finite element method. The results demonstrated strong consistency between experimental and numerical results, validating the pipeline's structural integrity. Opacic [4] has stated that numerous parameters strongly influence the assurance of safe operation of pressure vessels with unacceptable defects in welded joints. The most important factor is that not only the size of the defect is important for ensuring the integrity and operation of the vessel, but also the location of the defect. If a crack has occurred prior to exploitation, the integrity of the welded component must be determined at the location where it is most likely to occur [5].

The areas most susceptible to cracking are areas of geometric discontinuity, such as the connection of a cylinder with a nozzle or the connection of a cylinder with a hemispherical cap, where stress concentration occurs. At such locations, the stress field varies along the axial and circumferential directions, depending on the load under consideration [6]. To assess the criticality of surface cracks in cylindrical pressure vessels, a standard procedure is used that contains formulae for calculating the stress intensity factor. A component with a crack requires an assessment of the load-bearing capacity based on the determination of the fracture mechanics parameters and the plastic collapse load [7]. According to the criterion for ductile materials, the load must be less than the plastic collapse load. This reduces the problem of structural integrity to a two-parameter analysis in which the fracture mechanics criterion and the plastic collapse criterion must be fulfilled. The Failure Assessment Diagram (FAD) [8] is used for such assessments. The plastic collapse loads of defective welded components can be determined by an elastic-perfectly plastic FE analysis, whereby the load is increased until convergence is no longer possible. The collapse loads are important inputs to damage assessment methods, not only because they provide a limit load, but also because they are useful in predicting the response at lower loads, as demonstrated by the FAD approach [9].

Reference [10] described the determination of the effect of the detected flaw, modelling a partial torus of 8 mm length and 1 mm diameter concentrically arranged along the length of the weld. The flaw was modelled as it was detected, namely in a rounded shape. After analysing the results, it was found that the maximum equivalent stress does not exceed the yield strength of the material at elevated temperature, indicating that there is no plastic deformation under operating pressure loading. Furthermore, there is no significant stress concentration or significant change in stress distribution

During inspection of one such set of nozzle connections to header welded joints, a metalurgical flaw appeared and was manifested as rounded inclusion in wider area of the weld root. This flaw was found in most of the connections due to the same weld preparation and geometry, which facilitates the retention of slag in the weld root. Since any imperfections in the weld are undesirable and could pose a risk to the structure, this investigation focused on the behaviour of the weld between the head and nozzle and, above all, on the influence of such a flaw on the strength of the structure and its impact on reducing the load-bearing capacity of the pressure vessel.

2 PROBLEM DESCRIPTION AND METHODS

When inspecting the welded head-nozzle connections of a steam superheater, round inclusions were found at the root of the weld seam. The dimensions of the inclusion with a diameter of 1 mm and a length of 8 mm were determined from a ground macro pattern of the weld, which is shown in Fig. 1. The nozzles attached to the head had an outer diameter of 57 mm and a wall thickness of 6,3 mm, while the outer diameter of the head was 245,5 mm with a wall thickness of 30 mm. The entire system was to be designed for an operating pressure of 10,2 MPa. The finite element method was used with the finite element analysis software ASNYS to analyse the effect of the crack on the load capacity of the pressure vessel.

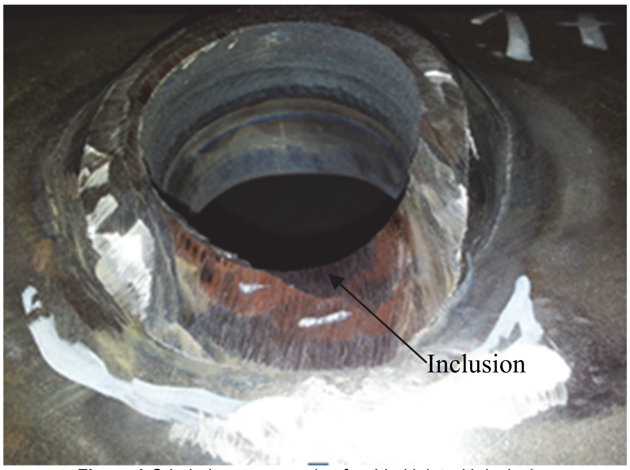

Figure 1 Grinded macro-sample of welded joint with inclusion

The location where the flaw occurred is obviously not susceptible to this type of defect. The stress analysis performed revealed that the most critical stress location is between the two welded nozzles. The situation could be worse if it were a sharp crack instead of a rounded inclusion flaw, as was considered in [10]. In this case, the inclusion is treated as a sharp elliptical crack and is located in a critical area, between two welded nozzles. An analysis was carried out with a sharp elliptical crack whose dimensions are $c = 4\,\text{mm}$ and $a = 0,5\,\text{mm}$, where c is the major radius and a is the minor radius. This crack is the largest flaw detected in the weld. The crack is located at the point of highest stress concentration, which is between the two welded nozzles (Fig. 3). The area around the crack was meshed with a finer mesh to improve the convergence of the numerical solutions. The crack is orientated in the sense that it opens (MODE I), i.e. the crack opens perpendicular to the local plane of the crack surface [11]. This symmetrical crack opening causes the maximum normal stress σ_1.

In addition to the case with a sharp crack in the critical location, the analysis takes into account the worst-case scenario, in which a flaw can spread over the entire length of the weld root. This extreme scenario was modelled using a flaw in the form of a full torus embedded in the fillet weld and positioned according to the analysis of the ground macrosample of the weld (Fig. 4). This comprehensive investigation allowed the potential impact of such a widespread flaw on the structural integrity of the weld to be assessed.

For the finite element analysis, 3D models were used that were meshed with tetrahedral finite elements, where the element size was 8 mm and the mesh was finer around the crack location (Fig. 2a). The finite element mesh consists of 566909 nodes and 407617 elements. The size of the elements is determined by a mesh sensitivity analysis of the results.

The material used is 16Mo3, which has a yield strength $\left(R_{\text{p0,2}} = 167\ \text{MPa}\right)$ according to EN 10273 at the operating temperature of the steam superheater $\left(\vartheta_r = 312°\text{C}\right)$. A steady-state thermal condition was considered in the FEM model, and the influence of temperature on the calculation results was taken into consideration by applying material properties for the operating temperature. In these analyses, the yield strength of the material in both the heat-affected zone and the welded joint, was assumed to correspond to the yield strength of the base material at the specified temperature. An elastic-perfectly-plastic material model was used. The loads applied in the analysis are static. The axial forces at the end of pipes were calculated based on the internal pressure, corresponding to the component's cross section. Axial forces values were proportionally increased with the increase of the internal pressure. The boundary conditions (displacements and loads) are defined as limitations of the axial displacements and forces at one end of the head and the internal pressure (Fig. 2b).

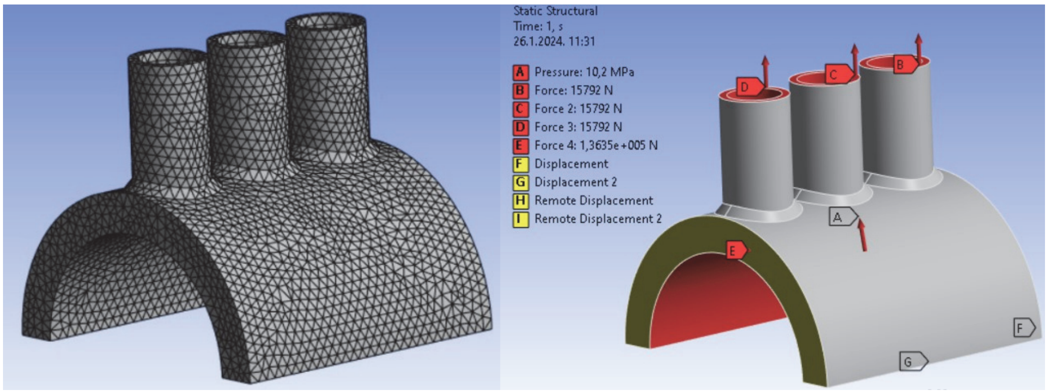

Figure 2 a) Finite element mesh, **b)** Boundary conditions and loads [10]

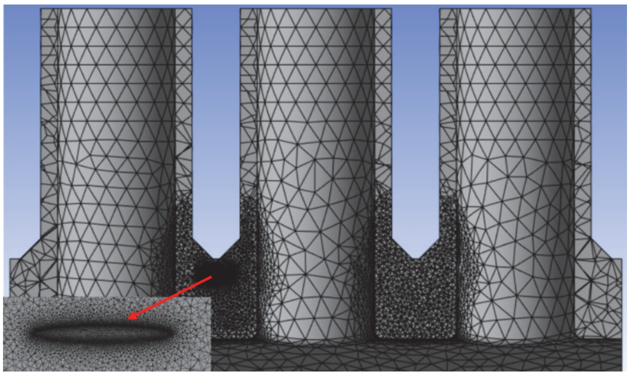

Figure 3 Model with sharp elliptical crack

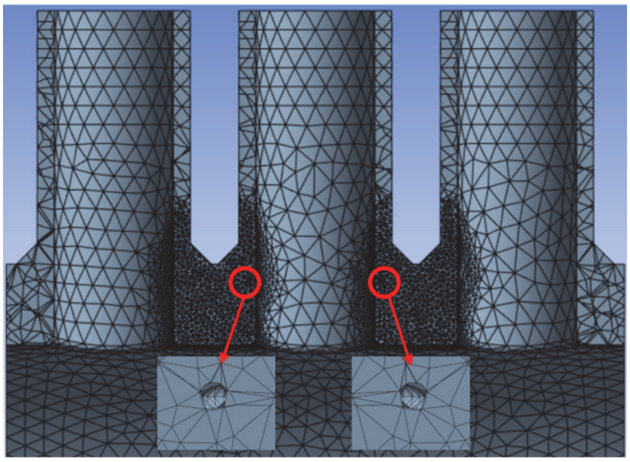

Figure 4 Model with torus shape flaw along whole circumference of weld

3 RESULTS

3.1 Limit Pressure

A numerical analysis was carried out to determine the plastic limit load by gradually increasing the internal pressure. The plastic limit load is defined as the pressure that the structure made of elastic-perfectly plastic material can withstand before failure. Von Mises stress distribution obtained from the numerical analysis is shown in the following figures. The area around the elliptical crack between the two nozzles is shown, in which the material has yielded over the entire area of the component. The grey colour shows the stress values that exceed the yield strength of the material at a temperature of 312 °C $\left(R_{\text{p0,2}} = 167\,\text{MPa}\right)$. The yielding of the material through the entire section occurs at an internal pressure of $p_{y,c} = 26,22$ MPa, for component with a sharp elliptical crack and $p_{y,t} = 24,1$ MPa, for component with a torus-shaped flaw. Fig. 5 shows the distribution of von Mises equivalent stress with increased internal pressure above the operating pressure of the steam superheater.

3.2 Stress Intensity Factor

Ansys Mechanical Fracture Mechanics [12] was used to determine the stress intensity factor using a fracture tool. The material was modelled with a linear elastic behaviour and obtained stress distribution is shown in Fig. 6. The stress intensity factor (SIF) values were also calculated for an elliptical crack located between two welded nozzles.

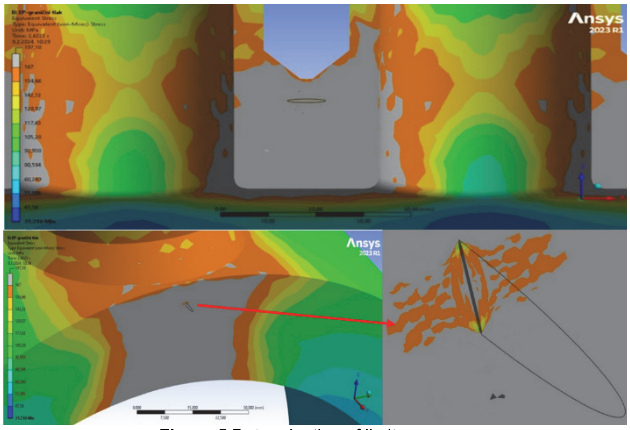

Figure 5 Determination of limit pressure

The maximum SIF value was $K_{\text{I}} = 2,173\,\text{MPam}^{1/2}$, and the minimum SIF value was $K_{\text{I}} = 0,861\,\text{MPam}^{1/2}$.

3.3 Fracture Toughness - K_{mat}

The fracture toughness K_{mat} is a property of metallic materials and depends on the temperature and the rate of loading. It is most commonly determined by the experimental method according to the American standard ASTM E399. To assess the integrity of components without experimental measurement of fracture toughness, correlation equations are used between the Charpy impact toughness and the standard measured fracture toughness. This is particularly important for components that have been in service for a long time and for which no fracture toughness data is available. A useful guideline in these situations is the empirical relationship between the material's resistance to crack growth and the impact energy (*KV*). A correlation expression was proposed as part of the SINTAP procedure [13]:

$$K_{\text{mat}} = \left[\left(12 \cdot \sqrt{KV} - 20\right) \cdot \left(\frac{25}{B}\right)^{0,25}\right] + 20 \tag{1}$$

where: *KV*, J - impact energy; *B*, mm - wall thickness of pipe.

For the material 16Mo3, the value of KV is 40 J (20°C) while the wall thickness of the pipe (*B*) at the point where the crack occurred is 25 mm. According to the above equation, $K_{\text{mat}} = 75,89\,\text{MPam}^{1/2}$.

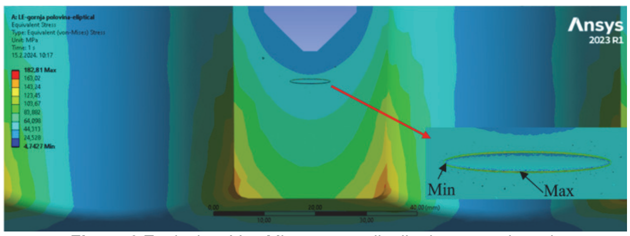

Figure 6 Equivalent Von Mises stress distribution around crack

3.4 SINTAP Procedure

The SINTAP procedure is used to evaluate the structural integrity of the welded head and nozzle structure with respect to the intensity of the load. The application of linear elastic fracture mechanics to brittle materials is based on the stress intensity factor during crack opening of the first mode $\left(K_{\mathrm{I}} < K_{\mathrm{mat}}\right)$. If the material is tough or ductile, fractures will occur due to plastic collapse. Plastic collapse can be avoided if the load p is less than the collapse load p_c ($p < p_c$). The integrity of structural components at risk of fracture can be assessed using a fracture assessment diagram (FAD). The fracture assessment is reduced to a two-parameter analysis that includes the criteria of fracture mechanics and plastic collapse. The values of the crack dimensions were defined and the limit pressure was determined with the stress intensity factor in order to be able to use the SINTAP assessment [14].

The ratio between K_{I} and K_{mat} is defined by the y - axis of the fracture estimation diagram using the following equation:

$$K_r = \frac{K_{\mathrm{I}}}{K_{\mathrm{mat}}} \tag{2}$$

where K_r is the ratio between K_{I} and K_{mat}. The maximum K_{I} was used from calculation from Section 3.2, which is $K_{\mathrm{I}} = 2{,}173\,\mathrm{MPam}^{1/2}$. The value of K_{mat} was calculated according to Eq. (1).

3.5 Plastic Collapse Limit for a Component with Torus-Shaped Flaw

The plastic collapse limit is determined also as the limit load for a plastic material according to the expression:

$$\sigma_f = \frac{R_{\mathrm{p0,2}} + R_{\mathrm{m}}}{2} \tag{4}$$

where σ_f is the plastic collapse limit, $R_{\mathrm{p0,2}}$ and R_{m} are the yield and the tensile strength of the material 16Mo3. At operating temperature $\left(\vartheta_r = 312\ °\mathrm{C}\right)$ $R_{\mathrm{p0,2}} = 167\,\mathrm{MPa}$ and $R_{\mathrm{m}} = 417\ \mathrm{MPa}$. Therefore, the plastic collapse limit can be taken as $\sigma_f = 292\,\mathrm{MPa}$. Component failure occurs when the stresses in the entire cross-section of the component exceed the plastic collapse limit. After the calculation has been conducted, component failure occurs at an internal pressure of $p_{c,t} = 41{,}67\,\mathrm{MPa}$, which is plastic collapse load for component with torus-shaped flaw.

Fig. 8 shows the distribution of the equivalent von Mises stress with increased internal pressure on 41,67 MPa for a welded joint with a torus-shaped flaw along the entire circumference of the weld root with a diameter of 1 mm.

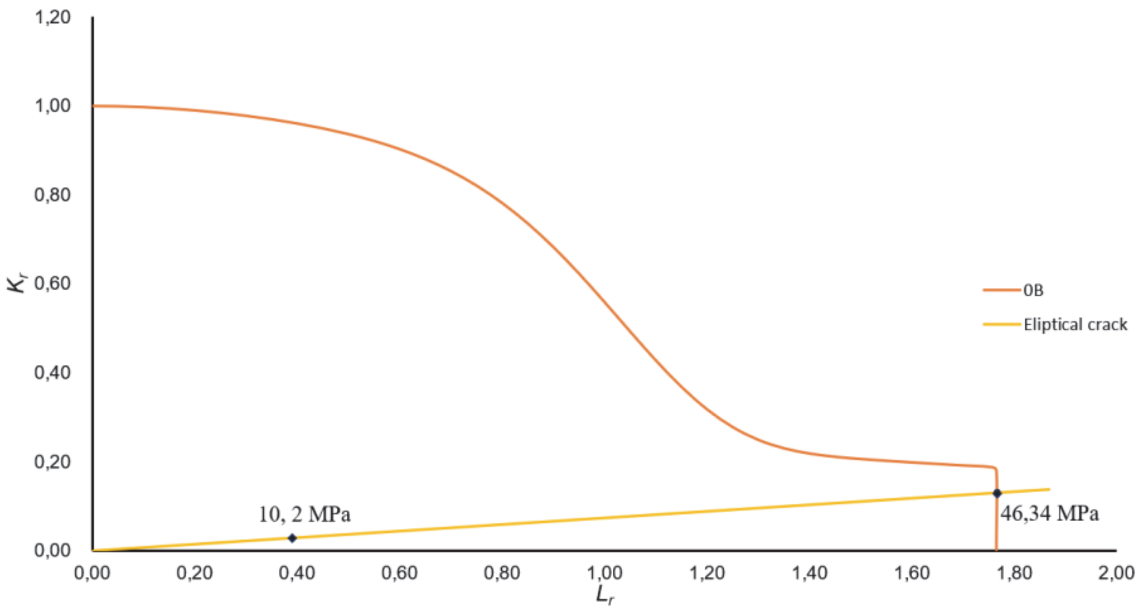

Figure 7 FAD diagram for an elliptical crack between two nozzles

The ratio between p and p_y is defined on the x - axis of the fracture assessment diagram using the following equation:

$$L_r = \frac{p}{p_y} \tag{3}$$

where L_r is ligament yielding parameter. The limit pressure p_y used for the sharp elliptical crack, is $p_{y,c} = 26{,}22\ \mathrm{MPa}$. For the value of p, the operating value of internal pressure of 10,2 MPa was used.

Fig.7 shows the position of the operating point for an elliptical crack located between two nozzles. The figure clearly shows that the operating point (10,2 MPa) is below the FAD curve, which means that it is in the safe zone. It can be clearly seen that the internal pressure at the yield point does not lead to failure, and with increase of operating pressure the failure could only occur when the plastic collapse load is reached. The FAD diagram shows that the point at which failure occurs lies on the FAD curve plastic collapse cutoff. The value of possible failure load for component with a sharp elliptical crack can be determined as $p_{\mathrm{c,c}} = 46{,}34$ MPa.

The grey colour indicates stresses that exceed the plastic collapse limit.

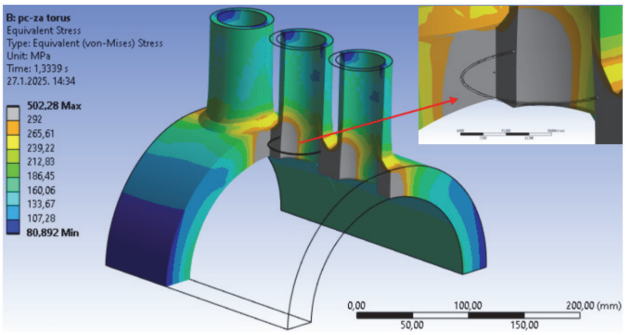

Figure 8 Model with a full torus-shaped flaw

Comparing the value with $p_{c,c}$ = 46,34 MPa, read from the FAD diagram, a possibility for a component failure is higher for torus-shaped flaw, because plastic collapse occurs at lower load.

4 CONCLUSION

In this paper, two models of the welded joint were modelled: a model assuming that the defect is present along the entire length of the weld in the form of a torus and a model with a sharp elliptical crack between two welding nozzles. The numerical analyses were carried out using the finite element method. From the results obtained, it can be concluded that the crack between the two nozzles does not significantly affect the static strength of the structure, as the stresses around the elliptical crack do not exceed the yield point of the material. After applying the SINTAP analysis, it was found that the structure can withstand an internal pressure of 10,2 MPa without failing, as the operating point is within the safe range of the FAD diagram.

When considering a highly plastic material, it is important to note that damage can primarily be caused by plastic collapse and not necessarily by unstable crack propagation. Component failure due to plastic collapse occurs when the internal pressure of the steam superheater would reach a value of plastic collapse load. This aspect is crucial for understanding the material behaviour under load and for predicting the performance and durability of steam superheater. The plasticity of the material often plays a key role in damage processes and can be of paramount importance when assessing the safety and reliability of structures or products.

Residual stresses that occur during welding of the head and nozzle can be considered for further research. An analysis can also be carried out for non-matched materials, taking into account the influence of the additional material. For further research, the limit size of the elliptical crack between the nozzles at which the structure does not fail can be determined.

5 REFERENCES

[1] Diamantoudis, A. & Labeas, G. (2005). Stress intensity factors of semi-elliptical surface cracks in pressure vessels by global-local finite element methodology. *Engineering Fracture Mechanics*, *72*(9), 1299-1312. https://doi.org/10.1016/j.engfracmech.2004.10.004

[2] Milanović, A., Sedmak, A., & Gnjatović, N. (2020). Application of fracture mechanics parameters to spherical storage tank integrity assessment. *Tehnicki vjesnik - Technical Gazette*, *27*(5), 1592-1596. https://doi.org/10.17559/TV-20191021140800

[3] Jeremić, L., Sedmak, A., Petrovski, B., Đorđević, B., & Sedmak, S. (2020). Structural integrity assessment of welded pipeline designed with reduced safety. *Tehnicki vjesnik - Technical Gazette*, *27*(5), 1461-1466. https://doi.org/10.17559/TV-20200413142538

[4] Opačić, M., Sedmak, A., Bakić, G., Milošević, N., & Milovanović, N. (2022). Application of advanced NDT methods to assess structural integrity of pressure vessel welded joints. *Procedia Structural Integrity*, *42*, 1185-1189. https://doi.org/10.1016/j.prostr.2022.12.151

[5] Kozak, D., Damjanović, D., & Katinić, M. (2016). Integrity assessment of the butt weld joint with defect according to EN ISO 6520-1, series 400. *Structural Integrity and Life.*

[6] Brighenti, R. (2000). Surface cracks in shells under different hoop stress distributions. *International Journal of Pressure Vessels and Piping*, *77*, 503-509. https://doi.org/10.1016/S0308-0161(00)00051-X

[7] Cicero, S., Gutiérrez-Solana, F., & Álvarez, J. (2008). Structural integrity assessment of components subjected to low constraint conditions. *Engineering Fracture Mechanics*, *75*(10), 3038-3059. https://doi.org/10.1016/j.engfracmech.2007.12.013

[8] Ainsworth, R. A. (1984). The assessment of defects in structures of strain hardening material. *Engineering Fracture Mechanics*, *19*(4), 633-642. https://doi.org/10.1016/0013-7944(84)90096-1

[9] Ainsworth, R. A., Zerbst, U., & Schwalbe, K. H. (2003). An overview of failure assessment methods in codes and standards. *Volume 7: Practical Failure Assessment Methods.* https://doi.org/10.1016/B0-08-043749-4/07053-1

[10] Konjatić, P., Kozak, D., Samardžić, I., & Despotović, B. (2013). Influence of technological flaw occurred during welding of header-nozzle joint using SAW welding process on the strength of welded joint. *Design, Fabrication and Economy of Metal Structures*, 483-488. https://doi.org/10.1007/978-3-642-36691-8_73

[11] Rice, J. R. (1968). Mathematical analysis in the mechanics of fracture. *Fracture: An advanced treatise*, *2*, 191-311.

[12] ANSYS, Inc. (2022). *Ansys Mechanical User's Guide, Release 2022.*

[13] Wallin, K. (1993). Recommendations for the application of fracture toughness data for structural integrity analysis. *Proceedings of the Joint IAEA/NEA International Specialists' Meeting on Fracture Mechanics Verification by Large Scale Testing*, 465-494.

[14] Jeyakumar, M. & Christopher, T. (2013). Defect assessment of welded specimen considering weld-induced residual stresses using SINTAP procedure and FEA. *Transactions of Nonferrous Metals Society of China*, *23*(5), 1452-1458. https://doi.org/10.1016/S1003-6326(13)62616-3

Contact information:

Ivan DUNĐER, mag. ing. mech.,
(Corresponding author)
University of Slavonski Brod,
Mechanical Engineering Faculty in Slavonski Brod,
Trg Ivane Brlić Mažuranić 2, HR-35000 Slavonski Brod, Croatia
E-mail: idunder@unisb.hr

Pejo KONJATIĆ, Full Professor, PhD
University of Slavonski Brod,
Mechanical Engineering Faculty in Slavonski Brod,
Trg Ivane Brlić Mažuranić 2, HR-35000 Slavonski Brod, Croatia
E-mail: pkonjatic@unisb.hr

Marko KATINIĆ, Associate Professor, PhD
University of Slavonski Brod,
Mechanical Engineering Faculty in Slavonski Brod,
Trg Ivane Brlić Mažuranić 2, HR-35000 Slavonski Brod, Croatia
E-mail: mkatinic@unisb.hr

Ivan GELO, Assistant Professor, PhD
University of Slavonski Brod,
Mechanical Engineering Faculty in Slavonski Brod,
Trg Ivane Brlić Mažuranić 2, HR-35000 Slavonski Brod, Croatia
E-mail: igelo@unisb.hr