Aleksandar M. Milovanović[1], Tamara Mijatović[2], Ljubica Diković[1], Ljiljana Trumbulović[1], Branko Drndarević[3]*

STRUCTURAL INTEGRITY ANALYSIS OF A CRACKED PRESSURE VESSEL

ANALIZA INTEGRITETA POSUDE POD PRITISKOM SA PRSLINOM

Originalni naučni rad / Original scientific paper
UDK /UDC:

Rad primljen / Paper received: 30.11.2021

Adresa autora / Author's address:
[1] Western Serbia Academy of Applied Studies, Užice, Serbia email: aleksandar.milovanovic@vpts.edu.rs
[2] University of Belgrade, Innovation Centre of the Faculty of Mechanical Engineering, Belgrade, Serbia
[3] Rad-Raso d.o.o., Užice, Serbia

Keywords
- finite element analysis
- structural integrity
- fracture mechanics
- crack
- pressure vessel

Ključne reči
- metoda konačnih elemenata
- integritet konstrukcije
- mehanika loma
- prslina
- oprema pod pritiskom

Abstract

Stress distribution, as obtained by finite element method (FEM), and corresponding fracture mechanics parameters, have been used for the integrity assessment of a pressure vessel. Once the stress distribution is known, the most critical area can be analysed with assumed crack to estimate its critical length according to fracture mechanics parameters and basic laws. Toward this end, the Failure Assessment Diagram (FAD) is used to calculate the crack length corresponding to the limit curve. As one can see, coordinates in FAD are the ratio of net- and critical stress (x axis, $S_r = S_n/S_c$), and ratio of stress intensity factor and its critical value (y axis, $K_r = K_i/K_{Ic}$). Once net stress and the stress intensity factor are known and their critical values, coordinates of a point corresponding to any crack length can be calculated and positioned in the FAD.

Izvod

Raspodela napona, dobijena metodom konačnih elemenata (MKE), i odgovarajući parametri mehanike loma, korišćeni su za procenu integriteta posude pod pritiskom. Kada je raspodela napona poznata, najkritičnija oblast se može analizirati sa pretpostavljenom prslinom da bi se procenila njena kritična dužina prema parametrima mehanike loma i osnovnim zakonima. U tom cilju, dijagram analize loma (FAD) se koristi za izračunavanje dužine prsline koja odgovara graničnoj krivoj. Kao što se vidi, koordinate u dijagramu su odnos napona u neto preseku i kritičnog napona (x osa, $S_r = S_n/S_c$) i odnos faktora intenziteta napona i njegove kritične vrednosti (y osa, $K_r = K_i/K_{Ic}$). Kada je poznat napon u neto preseku i faktor intenziteta napona, kao i njihove kritične vrednosti, koordinate tačke koje odgovaraju bilo kojoj dužini prsline mogu se izračunati i postaviti u FAD.

INTRODUCTION

The paper analyses an oil-storage tank (Fig. 1) of volume $V = 10000$ l, exposed to internal pressure $p = 78$ bar. The operating temperature of the oil-storage tank is 40 °C. Tank elements are: cylindrical shell and torispherical end made of material P355 GH /1/. Pipe fittings of the tank are made of material P265 GH /2/, and weld neck flanges are made of P245 GH /3/. The manhole flange and the blind manhole flange are made of material A105 LF2, /4/. The vertical support of the tank is made of S235 JR /5/.

The allowable stress of oil-storage tank elements is determined according to standard EN 13445-3:2021 /6/, using the following expression:

$$f = \min\left(\frac{R_m}{2.4}, \frac{R_{p0.2t}}{1.5}\right). \tag{1}$$

The allowable stresses for tank elements of P355 GH (EN 10028-2) are:
- cylindrical shell, $f = 212.50$ N/mm^2;
- torispherical end, $f = 212.50$ N/mm^2.

The allowable stress for tank elements of A105 (ASME/ASTM) is:
- flanges, $f = 165.33$ N/mm^2.

The allowable stress for tank elements of P265 GH (EN 10216-2) is:
- pipe fittings, $f = 170.83$ N/mm^2.

The calculation of the oil-storage tank elements is done according to standard EN 13445-3:2021 /6/, after which the wall thickness of the cylindrical shell is adopted, $e_n = 39$ mm, as well as the wall thickness of the torispheric end $e_s = 36$ mm.

Using finite element method, a stress analysis of a realistic 3D model of an oil-storage tank formed on the basis of technical documentation is performed. The analysed tank is assigned an appropriate finite element mesh, supports are installed and the tank is exposed to the calculated internal pressure.

By reviewing the obtained results, the zone of highest stress is established. The zone of maximal stress is additionally analysed by applying elastoplastic fracture mechanics parameters and introducing a non-existing fault in the welded joint. The impact of the introduced fault on the oil-storage tank structural integrity is inspected. By comparing the critical value of the stress intensity factor and calculated value of the stress intensity factor at which the depth of the introduced fault is varied, the critical depth value of the introduced fault is determined. The fracture analysis diagram graphically shows the change in depth of the introduced fault and its influence on the structural integrity of the oil-storage tank.

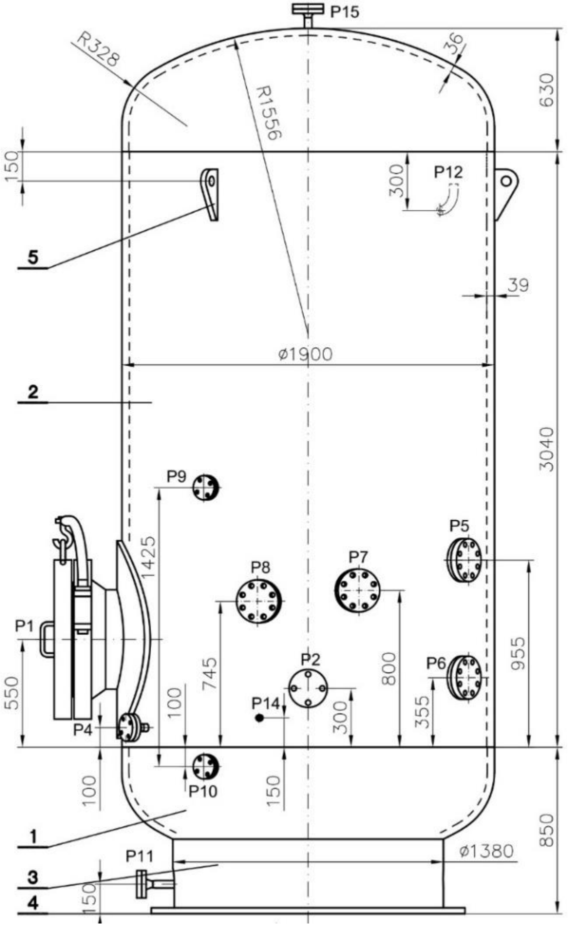
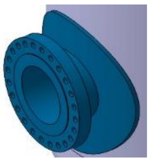


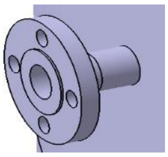
Figure 1. Oil-storage tank: 1- torispherical end, 2- cylindrical shell, 3- vertical support, 4- support plate, 5- crane transport eye.

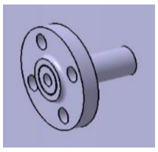
FINITE ELEMENT MODEL OF THE OIL-STORAGE TANK STRUCTURE

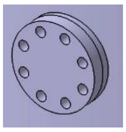
To form the finite element model of the oil-storage tank, all details are drawn in three-dimensional form on the basis of the original documentation (Figs. 1 and 2), /7/.

Figure 3 shows the automatically generated finite element mesh of uniform size of 24 mm. The finite element size adopted in this way guarantees deviations in stress values obtained for neighbouring nodes of the FE mesh not exceeding 5 %, which is satisfactory from the aspect of engineering practice.

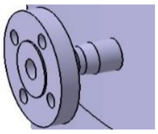

P1,

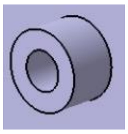

P2, P3


P9, P10


P5, P6, P7, P8


P12


P4, P11, P13


P14

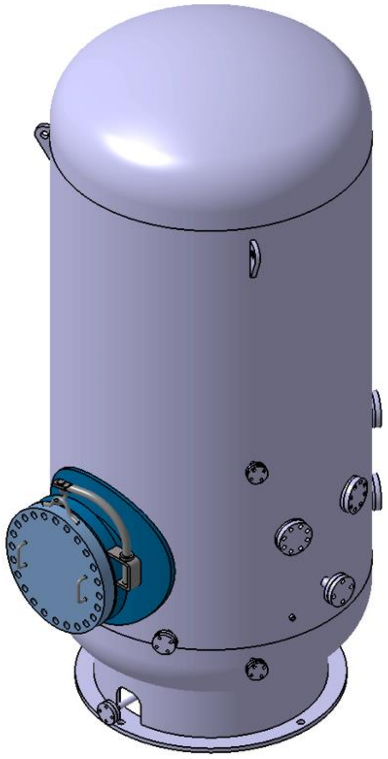


Figure 2. 3D model of oil-storage tank elements.

Figure 3. Finite element mesh.

The tank structure support is modelled by preventing all degrees of freedom in the nodes of finite element mesh in the zone of lower surface of support plate, Fig. 4.

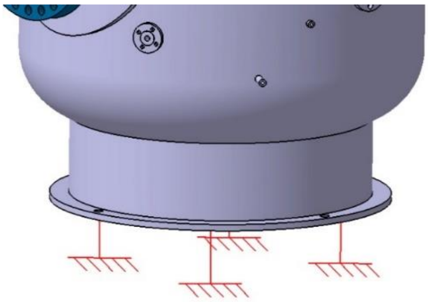


Figure 4. Defined support model.

The mesh consists of 683580 finite elements and 177855 nodes. Each node has 3 degrees of freedom so that the system has a total of 533565 unfactored equations. By reducing the system for given displacements, i.e. for 4890 equations in place of supports, the number of nonhomogeneous linear algebraic equations to be solved is 528675.

In matrix form /8/, the system of equations can be represented as:

$$\{F\}=[\mathrm{k}]\{\Delta\}\rightarrow\begin{Bmatrix}F_S\\F_R\end{Bmatrix}=\begin{bmatrix}K_{SS} & K_{SR}\\K_{RS} & K_{SS}\end{bmatrix}\begin{Bmatrix}\Delta_S\\\Delta_R\end{Bmatrix}. \qquad (2)$$

Since the displacement vector is in the places of supports,

$$\{\Delta_R\}=\{0\}. \qquad (3)$$

Unknown nodal displacements are obtained from matrix equation,

$$\{\Delta_S\}=[K_{ss}]^{-1}\{F_S\}, \qquad (4)$$

where: $\{\Delta_S\}$ is vector of unknown displacements; $\{F_S\}$ is the load vector; and $[K_{SS}]$ is the global stiffness matrix.

By determining the field of movement of the model, the system is statically identified. By using equations of displacement and deformation connections, as well as equations of deformation and stress, the stress state of the model is determined.

LOAD ANALYSIS

The oil-storage tank structure under design conditions is exposed to the effects of:

– net weight of equipment,
– calculated pressure (p = 78.0 bar).

Net weight of the equipment is automatically generated in the finite element analysis program, /7/, based on given material density (steel) and known volume of the model.

The calculated pressure is entered into the program for the analysis of stress-strain states using finite element method as a uniform pressure field acting on the inner surface of the oil-storage tank casing, and inner surface of the pipe fittings. A representation of the uniform pressure field input is given in Fig. 5.

STRESS-STRAIN OF STORAGE TANK STRUCTURE

The identification of the stress-strain state of the oil-storage tank structure is performed for calculated loads to be used later in structural integrity analysis, /8/. All displacements shown in displacement field images are given in mm. Images of the stress fields show values of comparative stresses obtained according to the hypothesis of the largest deformation work spent on the change of shape. Von Mises stress for a three-dimensional stress state is determined based on:

$$\sigma_u=\sqrt{\frac{1}{2}\left[(\sigma_x-\sigma_y)^2+(\sigma_y-\sigma_z)^2+(\sigma_z-\sigma_x)^2\right]+3(\tau_{xy}^2+\tau_{yz}^2+\tau_{zx}^2)} \qquad (5)$$

Stress values shown in stress field images are expressed in Pa. The appearance of the deformed structure is shown in Fig. 6. The largest displacement occurs in the zone of the upper torispherical end, and is 3.98 mm.

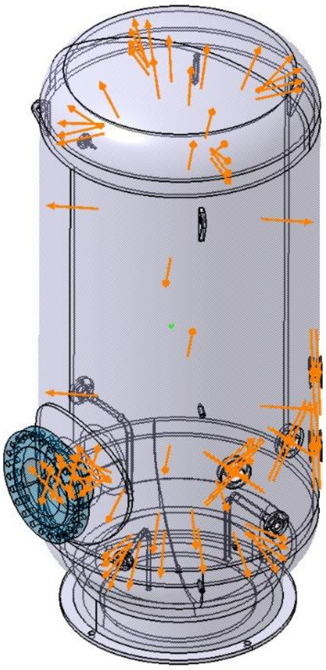


Figure 5. Display of uniform pressure field input.

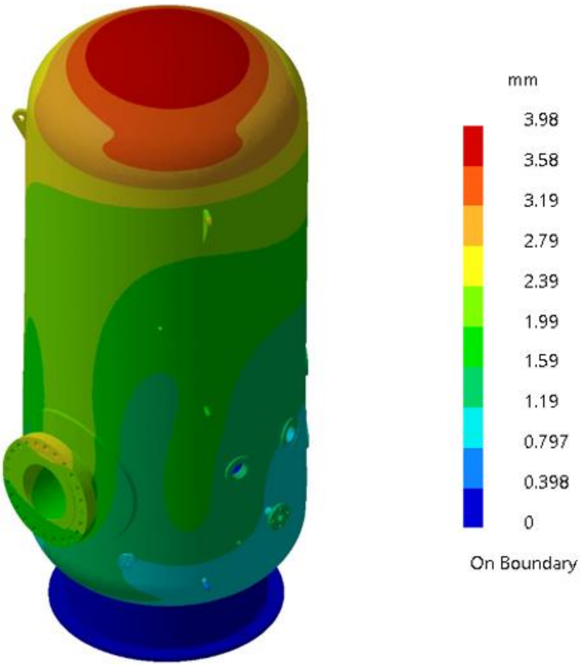


Figure 6. Displacement field of oil-storage tank structure.

The Von Mises stress is shown in Fig. 7 with a maximum (210 MPa) in the cylindrical tank shell in the zone of the fillet weld of the manhole reinforcement.

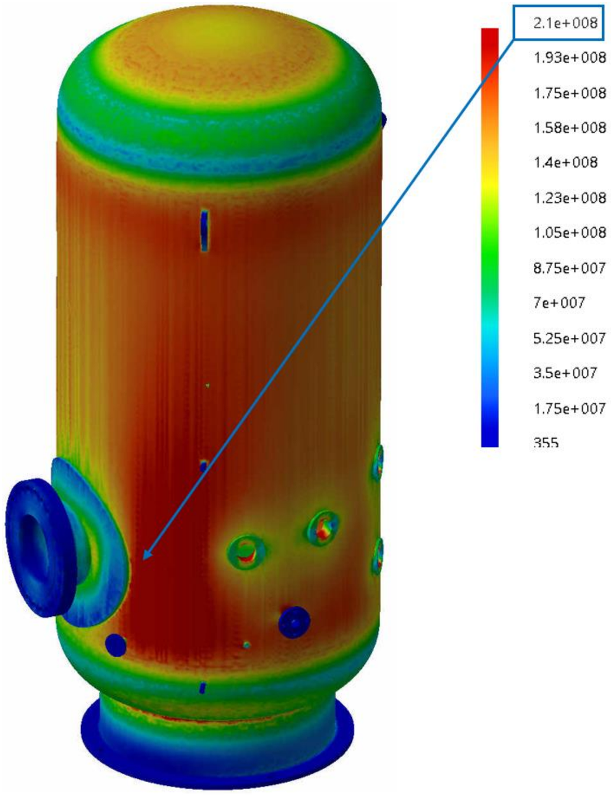


Figure 7. Stress field (N/m^2) of oil-storage tank structure.

STRUCTURAL INTEGRITY OF OIL-STORAGE TANK

The application of linear elastic fracture mechanics is based on the stress intensity factor K_I, /9/, which on the one hand represents the load and geometry of the structure, including the shape and size of the linear indication, and on the other hand its critical value, called fracture toughness K_{Ic}, /9/, a material property. Based on this interpretation of parameters of linear elastic fracture mechanics and Griffith energy criterion, /9/, structural integrity can be estimated by using the following relations:

- $K_I \leq K_{Ic}$ - integrity is not compromised,
- $K_I > K_{Ic}$ - integrity is compromised because brittle fracture is possible.

The methods of fracture mechanics can determine the range of depth of linear indication at which the integrity of the oil-storage tank structure will not be violated. In order to determine stress intensity factors, it is necessary to know the load and geometry, and the fracture toughness is thus determined by a conservative estimate of its value. Data required for detailed analysis are:

– vessel geometry (thickness t = 36 mm, mid radius R_{sr} = 932 mm),
– geometry of linear indication (depth a = ?),
– load (internal pressure p = 78 bar, residual stress σ_R = 0),
– fracture toughness 2500 MPa√mm, taken as min. value.

The stress intensity factor for longitudinal linear indications is determined based on the expression:

$$K_I = \left(\frac{pR_{sr}}{t} + \sigma_R\right)\sqrt{\pi a} \quad (\text{MPa}\sqrt{\text{mm}})\,. \tag{6}$$

The stress intensity factor for extensive linear indications is determined based on the expression:

$$K_I = \left(\frac{pR_{sr}}{2t} + \sigma_R\right)\sqrt{\pi a} \quad (\text{MPa}\sqrt{\text{mm}})\,. \tag{7}$$

Linear indications can be illustrated by failure assessment diagram, /9/. This approach has been recently used also for risk based structural integrity assessment, /10-13/. It includes evaluation of parameters K_r and S_r, defined as follows:

$$K_r = K_I / K_{Ic}\,. \tag{8}$$

$$S_r = \frac{\sigma_n}{\sigma_F}, \quad \sigma_n = \frac{pR_{sr}}{t}, \quad \text{and} \quad \sigma_F = \frac{R_{p0.2} + R_m}{2}\,. \tag{9}$$

The critical depth of linear indication is determined by increasing its value until the stress intensity factor K_I reaches its critical value K_{Ic}, as shown in Table 1. Corresponding points in the FAD are shown in Fig. 8.

Table 1. Critical value of linear indication depth.

crack depth, a (mm)	Stress intensity factor $K_I = \left(\frac{pR_{sr}}{t} + \sigma_R\right)\sqrt{\pi a}$ (MPa√mm)	Critical value of stress intensity factor K_{IC} (MPa√mm)
12	1859.79	2500
13	2019.9	2500
14	2191.43	2500
15	2376.36	2500
15.6	2494.7	2500

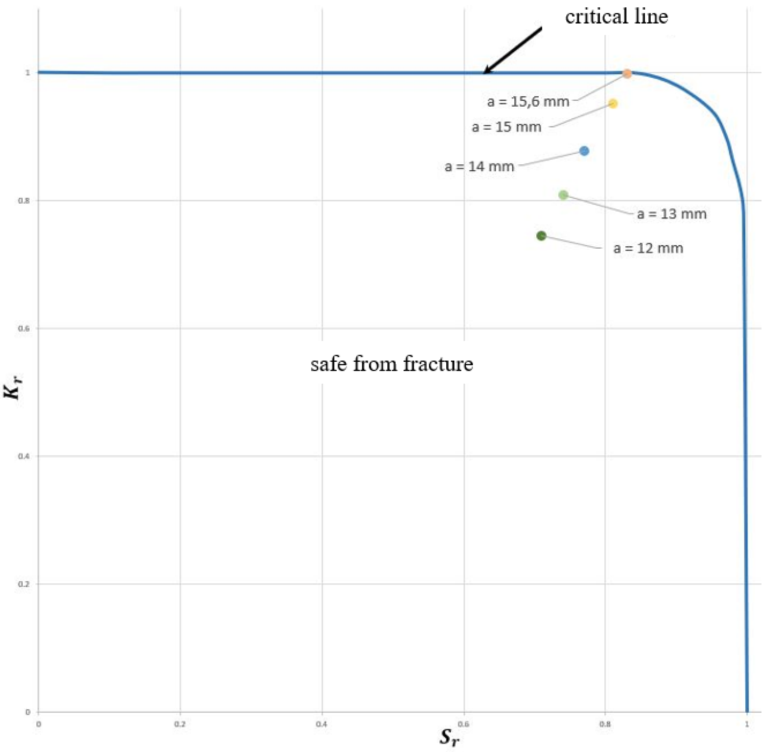


Figure 8. Fracture analysis diagram (FAD).

CONCLUSIONS

The analysed oil-storage tank, volume V = 10 000 l, is constructed according to applicable standards and regulations for pressure equipment. The calculation according to standard EN 13445-3: 2021 determines the minimum required wall thickness of the oil-storage tank elements, as based on which nominal values are adopted. According to the technical documentation, a 3D model of the oil-storage tank is created, as based on which a finite element model is formed, which is exposed to the action of internal pressure p = 78 bar. The analysis of the results obtained by applying the scientifically verified finite element method determined the zone of maximal stress (f_u = 210 MPa < f = 212.5 MPa). In order to analyse the integrity of the oil-storage tank structure, a non-existent welded joint fault is introduced into the maximum stress zone. The depth of the fault is considered to be of the size of the fault of the highest impact on structural integrity. The critical value of the stress intensity factor is K_{Ic} = 2500 MPa√mm. In order to determine critical fault depth, the stress intensity factor K_I is determined numerically, by increasing fault depth by 1 mm and a direct comparison with the value of critical stress intensity factor K_{Ic}. The presented method determined a critical value of fault depth of 15.6 mm that will not violate the integrity of the oil-storage tank, for which the stress intensity factor is K_I = 2494.7 MPa√mm and which is below the critical value of stress intensity factor K_{Ic} = 2500 MPa√mm.

REFERENCES

1. EN 10028-2:2017, Flat products made of steels for pressure purposes, 2017.
2. EN 10216-2:2013, Seamless steel tubes for pressure purposes - Technical delivery conditions, 2013.
3. EN 10222-2:2017, Steel forgings for pressure purposes, 2017.
4. ASME/ASTM A105, Specification for Carbon Steel Forgings for Piping Applications, 2010.
5. EN 10025-2:2004, Hot rolled products of structural steels, 2004.
6. EN 13445-3: 2021, Unfired pressure vessels. 2021.
7. Zamani, N.G., CATIA V5 FEA Tutorials Release 20, SDC Publications, 2011.
8. Sedmak, A. (2018), *Computational fracture mechanics: An overview from early efforts to recent achievements*, Fatig. Frac. Eng. Mater. Struct. 41(12): 2438-2474. doi: 10.1111/ffe.12912
9. Anderson, T.L., Fracture Mechanics - Fundamentals and Applications, 4th Ed., CRC Press, Taylor & Francis Group, Boca Raton, FL, 2017.
10. Jeremić, L., Sedmak, A., Milovanović, N., et al. (2021), *Assessment of integrity of pressure vessels for compressed air*, Struct. Integ. and Life, 21(1): 3-6.
11. Pilić, V., Mihajlović, V., Stanojević, P., et al. (2019), *Application of innovative risk assessment methodology for damage mechanisms identification on part of amine regeneration unit*, Struct. Integ. and Life, 19(1): 29-35.
12. Vučetić, I., Kirin, S., Vučetić, T., et al. (2018), *Risk analysis in the case of air storage tank failure at RHPP Bajina Bašta*, Struct. Integ. and Life, 18(1): 3-6.
13. Alnhoud, O.T., Shaqadan, A., Alkasasbeh, R. (2021), *Developing framework for safety analysis and risk assessment in construction site environment in Jordan*, Struct. Integ. and Life, 21(2): 135-145.
