The paper was presented at the Ninth Meeting
"New Trends in Fatigue and Fracture" (NT2F9)
Belgrade, Serbia, 12–14 October, 2009

Dražan Kozak, Pejo Konjatić, Franjo Matejiček, Darko Damjanović

WELD MISALIGNMENT INFLUENCE ON THE STRUCTURAL INTEGRITY OF CYLINDRICAL PRESSURE VESSEL

UTICAJ SMAKNUTOSTI ZAVARA NA INTEGRITET CILINDRIČNE POSUDE POD PRITISKOM

Original scientific paper
UDK /UDC: :621.791.05: 66-988
Paper received: 20.02.2010

Author's address:
J.J. Strossmayer University of Osijek, Mechanical Engineering Faculty in Slavonski Brod, Croatia

Keywords
- cylindrical shell
- weld misalignment
- stresses
- API 579
- finite element method
- surface circumferential crack
- failure assessment diagram (FAD)

Abstract

The influence of stresses resulting from weld misalignment in cylindrical shell circumferential weld joint on the shell integrity has been considered. The stresses have been estimated analytically by API recommended practice 579 procedure and calculated numerically by using the finite element method.

Surface circumferential crack has been assumed perpendicular to the principal stress at the location of maximum stress. Failure assessment procedure has been performed for the most dangerous situation (e = 8 mm) varying the crack depth to thickness ratio as a/t = 0.4; 0.45 and 0.5. Critical internal pressure values for these cases have been found by using of FAD diagrams.

Ključne reči
- cilindrična ljuska
- smaknutost zavarenog spoja
- naponi
- API 579
- metod konačnih elemenata
- površinska obimska prslina
- dijagram analize otkaza (FAD)

Izvod

Razmatra se uticaj napona nastalih zbog smaknutosti cilindričnih delova rezervoara pri zavarivanju na integritet posude pod pritiskom. Vrednosti napona procenjene su pomoću API 579 norme i izračunate numerički korišćenjem metode konačnih elemenata.

Pretpostavljena je površinska obimska prslina na mestu najveće glavnog napona, normalno na njen pravac. Izvedena je procedura pronalaženja pritiska otkaza za najopasniju situaciju (e = 8 mm) menjajući veličinu dubine prsline i debljine zida kao a/t = 0.4; 0.45; 0.5. Kritične vrednosti unutrašnjih pritisaka nađene su pomoću dijagrama ocene prihvatljivosti greške (FAD dijagram).

INTRODUCTION

Welded components are often places where flaws or damage can occur during their exploitation, but also in the fabrication stage. Different assessment procedures have been used to evaluate flaws encountered in pressure vessels and piping, but ASME Boiler and Pressure Vessels Code, Section VIII, Division 1 and Division 2, /1/, and API 579, /2/, are one of the most used. They were developed to provide guidance for conducting Fitness-for-Service (FFS) assessments. In this paper the fabrication tolerance given in Ref. /2/ for the centreline offset weld misalignment of circumferential joint in cylindrical pressure vessel which occurred during fabrication, /3/, has been critically considered by using finite element analysis results.

MISALIGNMENT DURING THE FABRICATION

Centreline offset weld misalignment in cylindrical shell circumferential weld joint has been described in Ref /2/. Diameters of the shell from one side to the other could be equal or unequal. Figure 1 presents the weld misalignment where diameters are equal ($D_1 = D_2$). The allowable centreline offset for $t \leq 19.1$ mm (3/4 in) can be calculated as, /2/:

$$e = \frac{t}{4} \tag{1}$$

where t is plate thickness.

In this paper weld misalignment of the circumferential joint (Category B, C and D) has occurred during the manufacture of the cylindrical pressure vessel with torispherical heads.

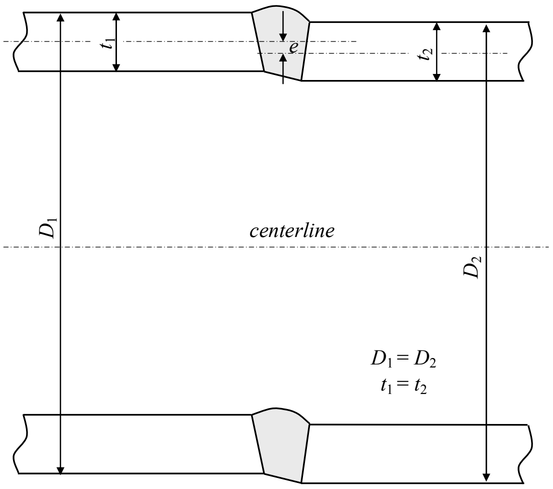

Figure 1. Centreline offset weld misalignment.
Slika 1. Geometrijsko odstupanje srednjih ploha delova cilindra nakon zavarivanja.

The outer diameter of the cylindrical pressure vessel is D_o = 2700 mm with plate thickness t = 14 mm. The detail of the circumferential misaligned weld joint with maximum amount of centreline offset is depicted in Figs. 2 and 3.

Figure 2. Weld misalignment in the vessel body.
Slika 2. Smaknutost kružnog zavara na spoju delova rezervoara.

Figure 3. Maximum weld misalignment.
Slika 3. Najveća smaknutost kružnog zavara.

The geometry of weld misalignment with maximum centreline offset e_{max} = 7.9 mm is presented in Fig. 4.

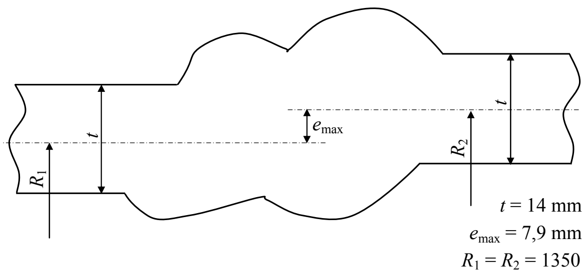

Figure 4. Geometry of weld misalignment.
Slika 4. Geometrija najveće smaknutosti zavarenog spoja.

According to Eq. (1) the allowable centreline offset can be calculated as:

$$e_{allowable} = \frac{t}{4} = \frac{14}{4} = 3.5 \text{ mm}$$

It is easy to note that measured maximum centreline offset is more than double regarding allowable value. This is why this investigation has to be conducted, trying to find this allowable centreline offset is too strict from the point of view of allowable stresses in the weldment.

FINITE ELEMENT ANALYSIS OF WELD MISALIGNMENT

Stress analysis of the cylindrical shell is done by using commercial code for finite element analysis – ANSYS 12.0, /4/, varying the values of centreline offset. The vessel material is S 355 J2 G3 structural steel with yield strength YS = 505 MPa and ultimate strength US = 608 MPa. Internal pressure of the vessel at working conditions is p_w = 1.64 MPa, but finite element simulation has been performed for hydrostatic test pressure p_h = 2.5 MPa. Axial-symmetric finite elements model with real geometry of the vessel is presented in Fig. 5.

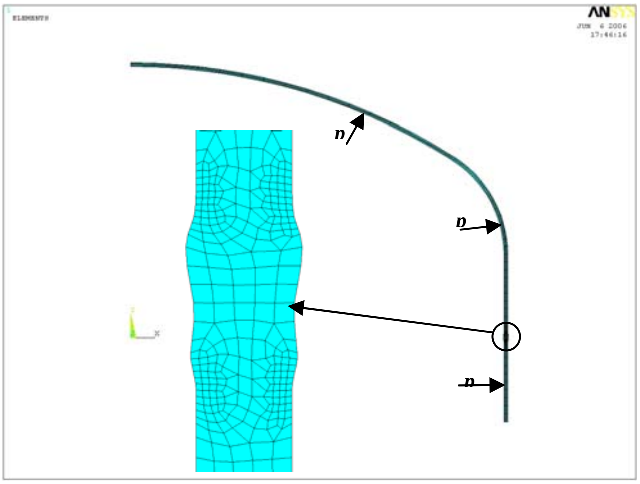

Figure 5. Finite element model of the vessel.
Slika 5. Model konačnih elemenata posude pod pritiskom.

Characteristic pressure vessel plane of axysymmetry has been discretized by isoparametric 8-node plane stress finite elements with the name PLANE 82 in ANSYS library. This element has a rectangular and triangular option, as well. Axial direction (Y) in Fig. 6 represents the axis of symmetry with X as axis in the radial direction. The mesh is rich in density at places where stress concentration is expected. Pressure is distributed on the inner surface of the vessel.

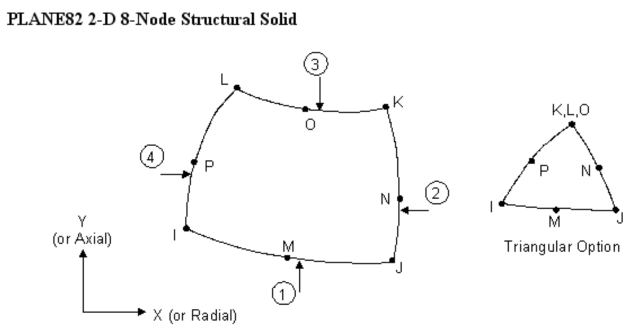

Figure 6. PLANE82 element from ANSYS Library, /4/.
Slika 6. Element PLANE82 iz ANSYS biblioteke, /4/.

Several finite element models are created to determine the influence of the centreline offset value on the stress distribution in the weldment: e = 2, 4, 6 and 8 mm (Fig. 7).

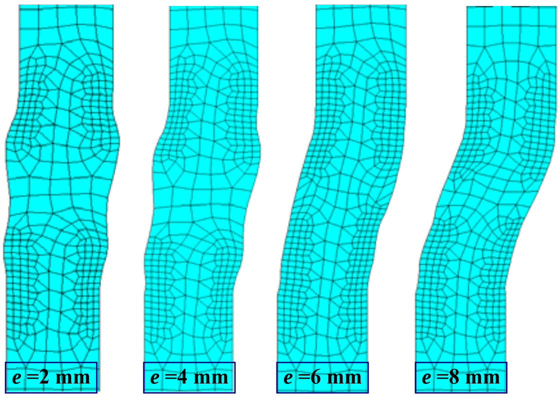

Figure 7. Finite element models for weldments of various misalignments.
Slika 7. Mreže konačnih elemenata za modele rezervoara s različitom smaknutošću zavara.

It is very important that welded toes should be rounded with the radius R_{min} = 20 mm to ensure larger bearing capacity of such performed circumferential welded joints, as presented in Fig. 8.

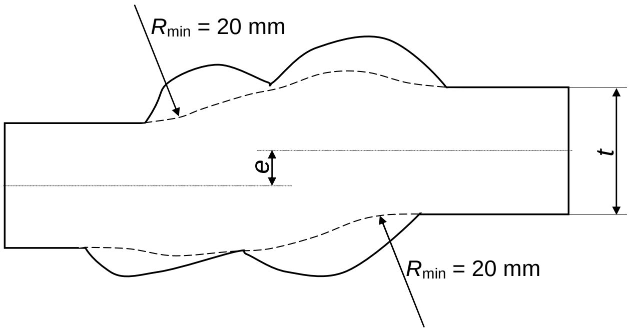

Figure 8. Roundness of the welded joint.
Slika 8. Zaobljenost zavarenog spoja.

COMPARISON OF STRESS RESULTS

Finite element analysis was performed for 5 geometries: first one is without weld misalignment (e = 0) and the next four with centreline offsets with the amount of 2, 4, 6 and 8 mm. There are hoop stresses in the vessel body (circular direction) σ_1 and axial stresses σ_2 as well, and equivalent von Mises stress σ_{eq}, which have been calculated. Minimal yield strength has been found for the weld material with the value YS = 479 MPa. It could be expected that for the maximum measured centreline offset value (e = 7.9 mm) corresponding to the finite element model with e = 8 mm, the most critical situation will be occur from the integrity point of view. The pressure vessel material is set as linear elastic-ideal plastic (Fig. 9), with yield strength YS = 505 MPa. The maximum principal stress distribution σ_1 and minimum principal stress σ_2 for this case are given in Figs. 10 and 11.

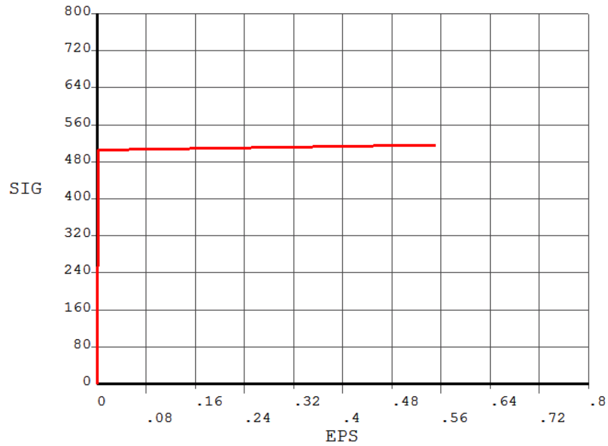

Figure 9. Linear elastic-ideal plastic material.
Slika 9. Linearno elastičan-idealno plastičan materijal.

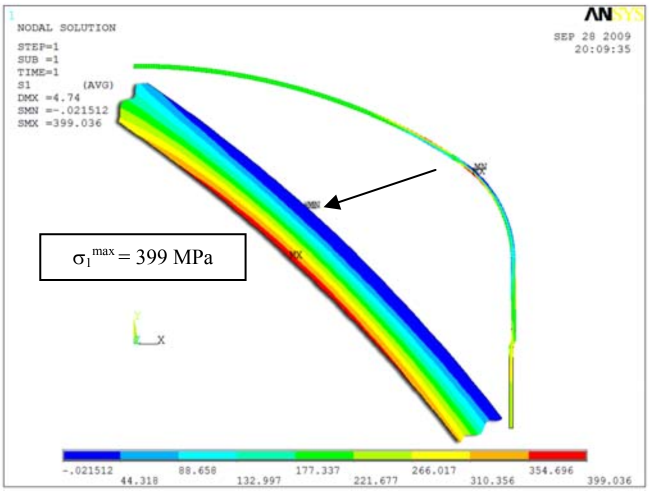

Figure 10. Principal stress σ_1 distribution (e = 8 mm).
Slika 10. Raspodela glavnog napona σ_1 (e = 8 mm).

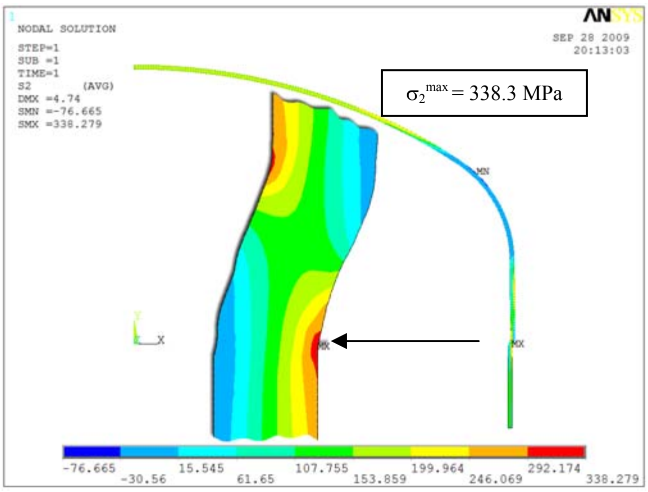

Figure 11. Principal stress σ_2 distribution (e = 8 mm).
Slika 11. Raspodela glavnog napona σ_2 (e = 8 mm).

It is easy to note from Figs. 10 and 11 that the place of minimum values of principal stress σ_1 and σ_2 is practically the same. But, maximum principal stress σ_1 is placed on the minimal meridian curvature of pressure vessel head, and maximum principal stress σ_2 is placed on misalignment. It is known from the membrane stress theory that in a cylinder, the hoop stress σ_h is double related to its longitudinal value σ_l. Figure 12 shows the directions of principal stresses in an ideally shaped cylinder.

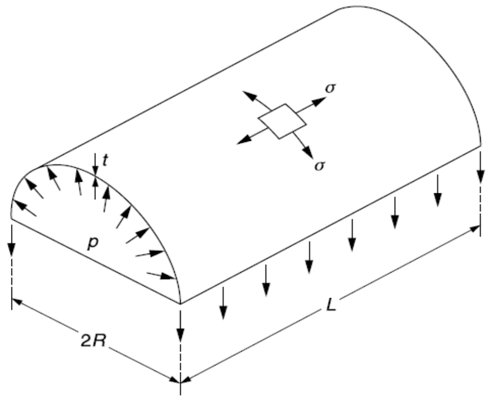

Figure 12. Stress directions in an cylindrical pressure vessel.
Slika 12. Pravci glavnih napona u cilindričnoj posudi pod pritiskom.

Hoop and longitudinal stresses from the membrane stress theory are known as:

$$\sigma_h = \sigma_1 = \frac{pR}{t} \tag{2}$$

$$\sigma_l = \sigma_2 = \frac{pR}{2t} \tag{3}$$

If we calculate the membrane stresses for our case:

$$\sigma_h = \sigma_1 = \frac{pR}{t} = \frac{2.5 \cdot 1350}{14} = 241.06 \text{ MPa}$$

$$\sigma_l = \sigma_2 = \frac{pR}{2t} = \frac{2.5 \cdot 1350}{2 \cdot 14} = 120.53 \text{ MPa}$$

It could be noted that strong bending effects exist. Principal stresses σ_1 are for any centreline offset greater than the theoretical value of σ_h. This is evidence that bending theory has to be applied in such a case. From the following chart (Fig. 13) it is obvious that bending affects at most principal stresses σ_2 with increase of weld misalignment.

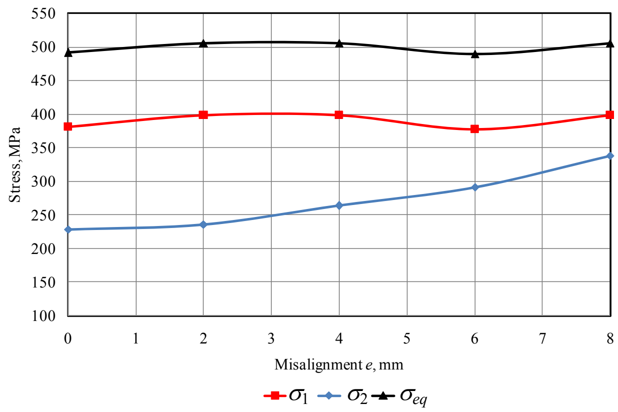

Figure 13. Stress variations due to different e values.
Slika 13. Promena napona sa smaknutošću e zavarenog spoja.

Generally, with the increase of centreline offset weld misalignment all relevant stresses simultaneously increase also. However, it can be calculated that even for the largest calculated principal stress σ_1^{max} that appears at maximum centreline offset (e = 8 mm), the yield strength value of the weld metal is not reached:

$$S = \frac{\sigma_1^{max}}{YS} \tag{4}$$

$$S = \frac{\sigma_1^{max}}{YS} = \frac{399}{479} = 0.83$$

This value could be considered as 17% of the reserve in material before its flowing. It could be considered as the inverse value of the minimum amount of safety factor:

$$SF = \frac{1}{S} \tag{5}$$

$$S = \frac{1}{0.83} = 1.2$$

POSTULATE SURFACE CIRCUMFERENTIAL CRACK AT THE LOCATION OF MAXIMUM STRESS

Special attention has been paid to the possibility that the surface circumferential crack could appear perpendicular to the maximal principal stress σ_2. Value of maximal principal stress σ_2 is ascertain at node 144, so assumption is that crack will appear perpendicular to the vector of maximal principal stress σ_2 at node 144, as shown on Fig. 14.

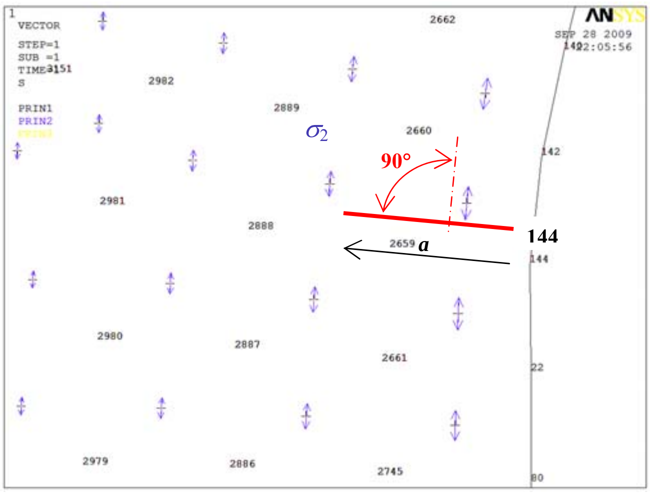

Figure 14. Crack will appear perpendicular to the vector of maximal principal stress σ_2.
Slika 14. Pojava prsline normalno na vektor najvećeg glavnog napona σ_2.

The crack is modelled as sharp in all cases of depth, with singular elements around the crack tip. The crack tip is defined as a place of stress concentration. It is necessary to calculate the stress and deformation distribution in the area of crack tip, because those values are essential for calculating fracture mechanics parameters afterward.

In case of shallower cracks, the critical area of material yielding is on the minimal meridian curvature of pressure vessel head, but not on the area around the crack, Fig. 15. Because of that, it is necessary to find the minimal crack depth for which the critical area of material yield in the area around the crack. Failure assessment procedure is performed, and it assures that the minimal crack depth is a = 5.65 mm, and limit pressure p = 3.48 MPa, Fig. 16.

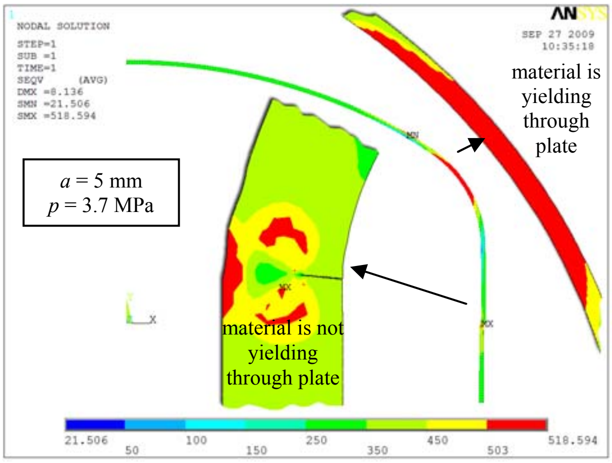

Figure 15. Material yielding in pressure vessel head, but not in the area around the crack.
Slika 15. Tečenje materijala u dancu posude, ali ne i u području oko prisutne prsline.

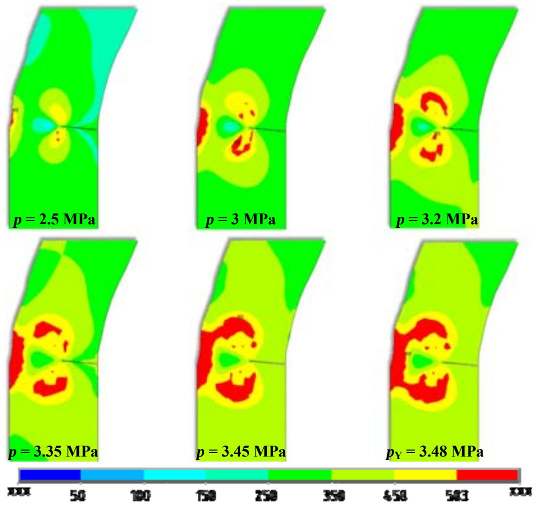

Figure 16. Material yield spread, a_{min} = 5.65 mm.
Slika 16. Tečenje materijala kroz debljinu zida, a_{min} = 5,65 mm.

Because of the need for limit pressure (p_Y) values for subsequent construction of the FAD diagram, they are calculated for specified examples by using finite element method. Failure assessment procedure has been performed for the most dangerous situation (e = 8 mm) varying the crack depth to thickness ratio as a/t = 0.4; 0.45; 0.5, and for the situation when misalignment e is 0 (e = 0 mm) and ratio a/t = 0.5. The analysis starts with crack of ratio a/t = 0.4 (Fig. 16), because this crack is ascertained as critical, when the area of material yield is located in the area around the crack. Limit pressure values for these cases have been found starting from hydro-test pressure (p_h = 2.5 MPa) up to the value that causes plastic yield through the ligament of the shell (Figs. 17 and 18).

Limit pressure values obtained with failure assessment procedure are shown in Table 1. Hereafter, critical internal pressure values for these cases will be found by using FAD diagrams.

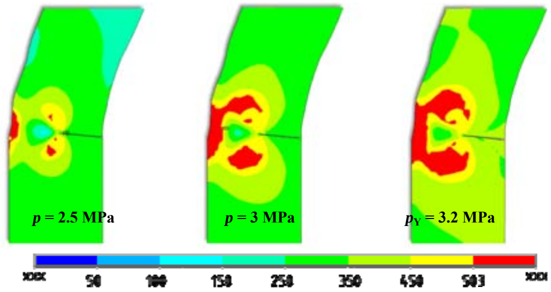

Figure 17. Material yield spread, a = 6.3 mm.
Slika 17. Tečenje materijala kroz debljinu zida, a = 6.3 mm.

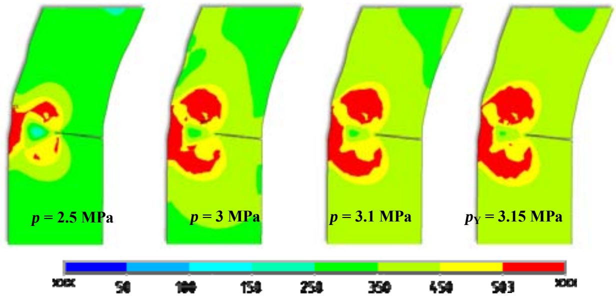

Figure 18. Material yield spread, a = 7 mm.
Slika 18. Tečenje materijala kroz debljinu zida, a = 7 mm.

Table 1. Values of limit pressure p_Y.
Tabela 1. Vrednosti graničnog pritiska p_Y.

a, mm		p_Y, MPa
5.65		3.48
6.3		3.2
7	With misalignment	3.15
	Without misalignment	3.7

With performed failure assessment procedures it is easy to conclude that limit pressure p_Y decreases with increase in crack depth a. But, even for crack depth (a = 7 mm) at the half of plate thickness (t = 14 mm) it is p_Y = 3.15 MPa, which is for 0.65 MPa greater than the hydro-test pressure (p_h = 2.5 MPa)!

THE INFLUENCE OF WELD MISALIGNMENT ON THE STRESS INTENSITY FACTOR VALUE

For calculating the stress intensity factor K_I as a local parameter defined on tip of the crack, as a function of load and crack geometry, there is no analytical term for an example as this one considered here. Therefore, the value of the stress intensity factor is calculated with finite element method from results of displacement of nodes of the crack.

A pressure vessel without misalignment was analysed for the purpose of comparison with a pressure vessel with misalignment (with an internal and an external crack). Compared is the influence of misalignment and the position of crack to the stress intensity factor value. For all cases, the pressure vessel with crack depth a = 7 mm (a/t = 0.5) was analysed. Internal pressure changes from pressure in working conditions (p_w = 1.64 MPa) up to p = 3.5 MPa, misalignment is e = 8 mm. Material is set as linear-elastic with Young's modulus of elasticity E = 206000 MPa, and Poisson's ratio ν = 0.3.

As the diagram displays (Fig. 19), the stress intensity factor values are changing by a linear principle in all cases. By increasing pressure values, the difference in the stress intensity factor in all cases is increases too. Therefore, the reason is obvious from the diagram why only the external crack is analysed. This is the case with maximum values of stress intensity factor, so this case is considered as the most critical.

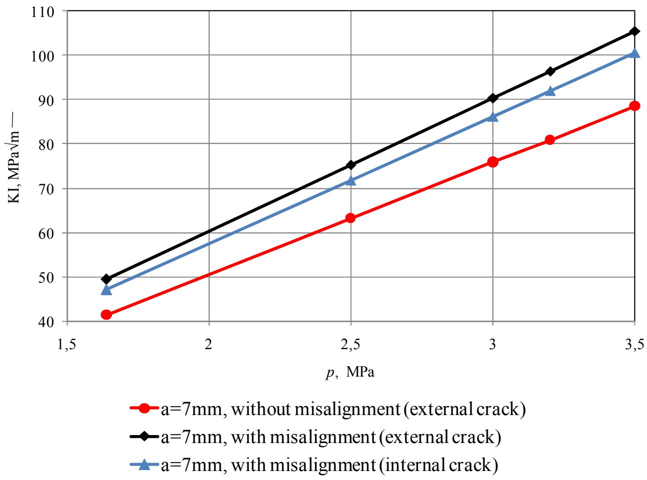

Figure 19. Stress intensity factor variation due to different pressure values ($a/t = 0.5$).
Slika 19. Promena iznosa faktora intenziteta napona u zavisnosti od unutrašnjeg pritiska ($a/t = 0.5$).

FAILURE ASSESSMENT DIAGRAM (FAD)

Because the properties of the material S 355 J2 G3 (toughness, yield and ultimate tensile strength) are known, it is necessary to use SINTAP Basic Level 1, /5/. In that case, the failure assessment diagram consists of three curves depicted by following equations (for material with Lüders plateau):

$$f(L_r)=\left(1+\frac{1}{2}L_r^2\right)^{-\frac{1}{2}} \quad \text{for } L_r<1 \tag{6}$$

$$f(1)=\left(\lambda+\frac{1}{2\lambda}\right)^{-\frac{1}{2}} \quad \text{for } L_r=1 \tag{7}$$

where λ is:

$$\lambda=1+\frac{0.375E}{\sigma_Y}\left(\frac{1-\sigma_Y}{1000}\right) \tag{8}$$

$$f(L_r)=f(1)\cdot(L_r)^{\frac{N-1}{2N}} \tag{9}$$

$$\text{for} \quad 1<L_r<L_r^{\max}=\frac{1}{2}\left(\frac{\sigma_Y+\sigma_U}{\sigma_Y}\right) \tag{10}$$

where N is:

$$N=0.3\left(1-\frac{\sigma_Y}{\sigma_U}\right) \tag{11}$$

The failure assessment diagram constructed from above equations is shown in Fig. 20.

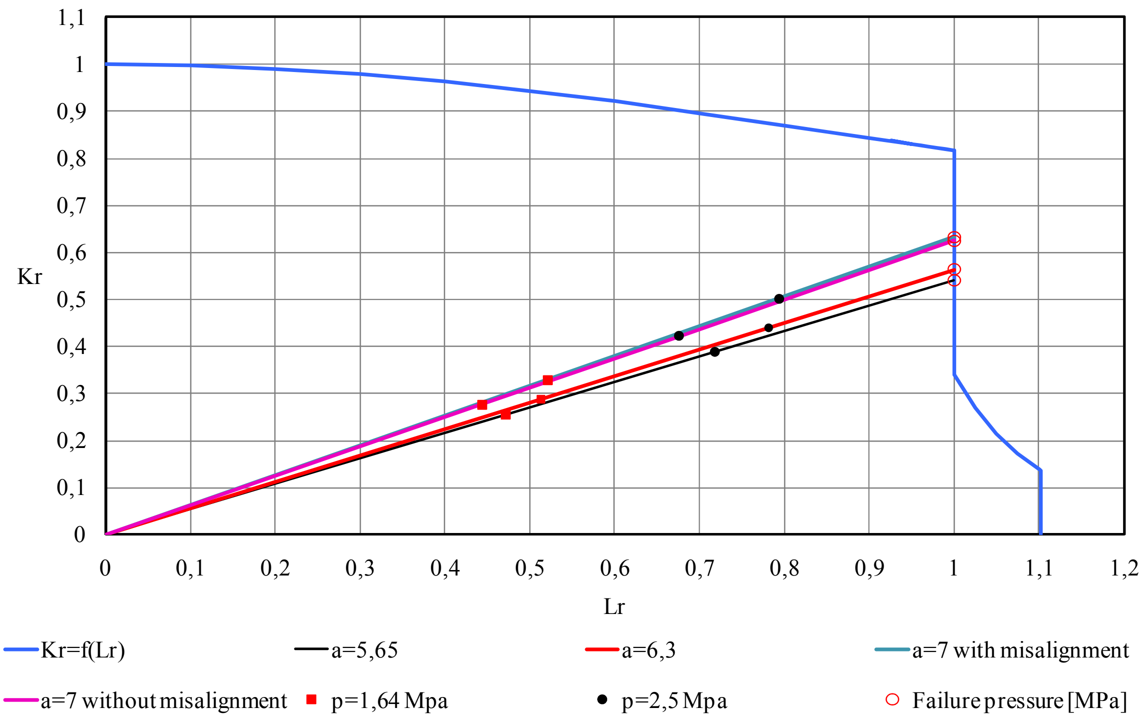

Figure 20. Failure assessment diagram (FAD diagram).
Slika 20. Dijagram ocene prihvatljivosti greške (FAD dijagram).

Loading paths in the FAD diagram are given for three crack depth values a = 5.65; 6.3 and 7 mm (with and without misalignment). It is obvious from the failure assessment diagram that all crack loading paths intersect the material curve in the point with L_r = 1, practically. This means that failure pressure of the cracked vessel is equal to the limit pressure value for the analysed cases. So, there is no need for calculating failure pressure from failure assessment diagram with the "backward method", because $L_r \cong 1$ for all cases! It should be noted that K_{mat} value has been calculated from the Charpy impact toughness value as $K_{mat} = 150$ MPa$\sqrt{\text{m}}$. Points are also specified on the FAD diagram that correspond to the working pressure (p_w = 1.64 MPa) and hydro-test pressure (p_h = 2.5 MPa), as well.

CONCLUSION

Influence of different centreline offset values by weld misalignment on the stress magnitude has been evaluated by finite element analysis and critically compared with the standard API 579. It could be concluded that recommended practice given in API 579 is conservative related to the results for principal stresses obtained by finite element method. Even double centreline offset values related to those which are allowable by standard could be accepted.

Special attention is paid to the possibility that the surface circumferential crack could appear perpendicular to the principal stress σ_2 at the location of maximum stress. Limit pressures are found by finite element analysis for three crack depths a/t = 0.4; 0.45; 0.5.

The corresponding FAD diagram has been constructed for the SINTAP Basic level 1. Failure pressures have been equal to limit pressure (all crack loading paths intersect the material curve in the point with $L_r = 1$).

For the most dangerous analysed case ($a = t/2 = 7$ mm and $e = 8$ mm), the limit pressure p_Y is 3.15 MPa, what is still by 0.65 MPa greater than the hydro-test pressure (p_h = 2.5 MPa) and almost double than the working pressure (p_w = 1.64 MPa).

REFERENCES

1. ASME Boiler and Pressure Vessels Code, Section VIII, Division 1 and Division 2, 2007.
2. API Recommended Practice 579, American Petroleum Institute, March 2000.
3. Technical report for Welded Vessels Ltd Slavonski Brod, 2006
4. ANSYS 12.0, Manual guide, 2009.
5. Kim, Y.-J., Comparison: Various levels of the SINTAP procedure for homogeneous structures, GKSS, 1998.