Martina Balać, Aleksandar Grbović, Aleksandar Petrović

NUMERICAL PREDICTIONS OF CRACK GROWTH IN A PRESSURE VESSEL WITH WELDED NOZZLES

NUMERIČKE PROCENE RASTA PRSLINE NA POSUDI POD PRITISKOM SA ZAVARENIM PRIKLJUČCIMA

Originalni naučni rad / Original scientific paper
UDK /UDC: 62-988-112.81:539.42
Rad primljen / Paper received: 20.05.2015

Adresa autora / Author's address:
University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia, email: agrbovic@mas.bg.ac.rs

Keywords
- XFEM
- 3D digital image correlation
- stress intensity factor
- crack growth
- nozzles

Abstract

Most structural elements that represent parts of equipment and machines in the process industry are subjected to strength calculations. Structural materials and welded joints typically contain flaws and microcracks, from which cracks are initiated. Exploitation conditions can lead to the occurrence of cracks, even in cases when there are no flaws in the material, typically at locations with stress concentration. Penetration of two cylinders is the most commonly encountered form of geometric discontinuity of cylindrical surfaces which results in stress concentration. On a pressure vessel with two nozzles of different geometries, critical areas (i.e. stress concentration areas) are determined by experimental 3D Digital Image Correlation (DIC) method. Then, the numerical analysis of the equivalent 3D model is performed and the obtained results are comparable to experimental values. Since the fatigue cracks are expected in the high stress areas, in one of them – next to the nozzle in the numerical model – a crack has initiated. Then, crack growth is simulated using extended finite element method (XFEM). The aim of this paper is to show that it is possible to predict the crack growth direction and critical length of the crack which can occur in the pressure vessel, based on values of stress intensity factors (SIFs) evaluated in the numerical simulation.

Ključne reči
- proširena metoda konačnih elemenata
- 3D digitalna korelacija slika
- faktor intenziteta napona
- rast prsline
- priključci

Izvod

Većina konstrukcionih elemenata koji predstavljaju delove opreme i aparata u procesnoj industriji podležu proračunima čvrstoće. Konstrukcijski materijali i zavareni spojevi najčešće sadrže greške i mikroprsline koje su začeci prslina. Uslovi eksploatacije mogu da dovedu do pojave prslina, čak i ako nema grešaka u materijalu i to najčešće na mestima koncentracije napona. Prodor dva cilindra je najčešći oblik diskontinuiteta geometrije cilindričnih površina, što kao posledicu ima pojavu koncentracije napona. U ovom radu je analizirano ponašanje jedne posude pod pritiskom na čiji omotač su zavarena 2 priključka različitih dimenzija. Eksperimentalnom 3D DIC metodom određena su kritična mesta (tj. mesta koncentracije napona), a zatim su numeričkim proračunima na ekvivalentnom 3D modelu dobijeni rezultati bliski eksperimentalno dobijenim vrednostima. S obzirom da je verovatnoća pojave prsline usled zamora najveća upravo u oblastima koncetracije napona, inicirana je prslina na numeričkom modelu koja je zatim širena primenom proširene metode konačnih elemenata (XFEM). Cilj ovog rada je da se pokaže da je na osnovu rezultata jedne ovakve numeričke simulacije, odnosno dobijenih vrednosti faktora intenziteta napona (SIFs), moguće predvideti smer rasta i kritičnu dužinu prsline nastale na posudi pod pritiskom.

INTRODUCTION

Pressure vessels are the most commonly used type of vessels in different industrial sectors, such as the nuclear, oil, petrochemical, and chemical. Considering that pressure vessels are usually subjected to different load types (static, dynamic, thermal, etc.), full attention is given to design, construction, testing and inspection of equipment. Failures usually occur in places of geometrical discontinuity. One of these discontinuities is the intersection of cylindrical shapes, whereas the problem frequently encountered in practice, is the influence of pipe nozzles on the stress and strain state of pressure vessels. Nozzles placed on the vessels are for different purposes, and they have their own structural characteristics as well as specific influences on the strain of the cylindrical shell. Maximum stress and strain values are exactly expected to occur at the junction of nozzle and cylindrical shell.

So far, research in the area of pressure vessels, i.e. measuring and determining stresses and strains has relied on analytical calculations, /1-3/, numerical calculations /4-7/ and experimental methods /8-9/. Limitations of current experimental methods include local measuring, i.e. obtaining measured values in a single point. By analysing a large number of pressure vessels, it is concluded that it is necessary to know material properties in critical areas, as well as their behaviour during exploitation, in order to assess the safety of one such welded structure.

The aim of this research is to show the influence of two nozzles on the stresses and strains of the vessel cylindrical shell, subjected to internal pressure, by experimental and numerical methods. In order to measure the whole strain field, a relatively new experimental method for three-dimensional optical strain analysis, based on Digital Image Correlation, /10, 11/, is used. Based on experimental values a numerical model of the vessel is developed. Extended Finite Element Method (XFEM) is chosen in order to analyse the behaviour in a specific zone of geometric discontinuities, as well as for the numerical simulation of the crack propagation in the pressure vessel.

EXPERIMENTAL METHOD

Experimental research is based on developing an experimental setup (model). Figure 1 shows the experimental setup used in this research.

One pressure vessel with two adjacent nozzles is designed and fabricated for this experimental study. Calculations and measurements are carried out and results are presented in this paper. The experiment is performed on a horizontal cylindrical pressure vessel (D = 378.4 mm, e_a = 1.5 mm and length 770 mm). Pressure vessel is made of X5Cr Ni 18 10 (EN 10088). Two nozzles, nozzle 1 DN50 (d_1 = 60.3 mm, e_1 = 2.9 mm) and nozzle 2 DN32 (d_2 = 42.4 mm, e_2 = 2.6 mm), are placed in the middle of the cylindrical shell, so that the longitudinal axis of the nozzle is perpendicular to the vessel's longitudinal axis. Minimal distance between two adjacent nozzle centres is calculated according to standard EN 13445-3. Nozzle height is selected in a way that ensures that there is no significant influence on stress and strain states. Two pipe nozzles are welded on one lid, made of the same material. One nozzle is used for connecting the manometer, whereas the other is used for connecting the tap that connects with the hose used for filling/emptying the vessels, as well as for the pump used to achieve overpressure. The vessel is tested on the pressure testing installation, subjected to internal pressure using water at the temperature of 20°C, Fig. 1.

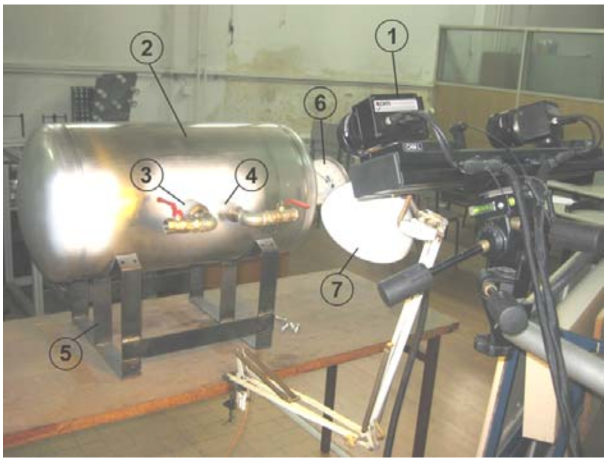


Figure 1. Experimental setup (1-cameras, 2–pressure vessel, 3, 4–nozzles 1-2, 5–metal support, 6-pressure gauge, 7–illumination).

Experimental investigation of strain field between two nozzles is conducted using 3D DIC method, /12/. Software package Aramis, along with the optical recording equipment, represents a system for analysis of three-dimensional strain, which successfully combines the accuracy of laboratory testing with the convenience of the finite element method.

The 3D system Aramis uses two digital cameras for full strain field measurement which are calibrated and prepared prior to measuring, and enables precise determining of critical areas, i.e. the highest strain values on cylindrical shell.

Parameters for basic Aramis system setup are: measuring volume 100×75, measuring distance 800 mm, camera angle 26º, calibration object CP20 90×72.

Area of the structure is of great importance during the measuring, hence before the measurements, the following important conditions must be fulfilled: the specimen surface must be prepared and free of grease and oil, the surface of the measured object needs to have a specific pattern in order to clearly define the pixels in camera images. In addition, the surface pattern must be matte, since reflected light causes poor contrast and difference in brightness between left and right camera, which may lead to inaccurate results. The measuring volume is adjusted based on the dimensions of the measuring field between nozzles. The surface of the measuring area is sprayed with white paint as base colour to form a thin uniform background. After the base colour dries, a stochastic pattern of black dots is sprayed. The measuring area is shown in Fig. 2

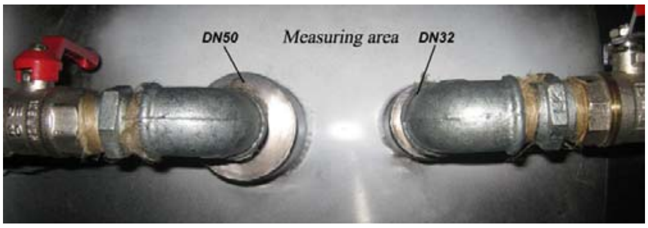


Figure 2. Measuring area.

Before using the system, it is necessary to adjust the sensor unit, adjust the angle between the lens focus and aperture. To ensure dimensional consistency of the measuring system, calibration is performed with the help of a calibration panel, and the entire system is calibrated. Once the measuring volume is successfully calibrated, experimental measuring can take place.

Applying the load as internal pressure and recording using a 3D system is performed manually. Vessel loading is applied by increasing pressure, according to the procedure defined in the EN 13445-3 standard.

According to EN 13445-3 standard, calculation had shown that the vessel's material would behave within its elasticity limits, up to pressure of 1 MPa, experimental results for full field strain distribution, between two closely welded nozzles, for 1.5 MPa internal pressure, are presented. Figure 3 shows the strain field between nozzles.

Three sections are placed in the measuring field 1, Fig. 3. Section 1 is positioned at the smallest distance between nozzles, sections 2 and 3 are positioned at 15 mm distance each from section 1. The highest Von Mises strain value is about 0.19% near the nozzle DN50. In Figure 4 the strain distribution of characteristic sections, depending on section length, is shown.

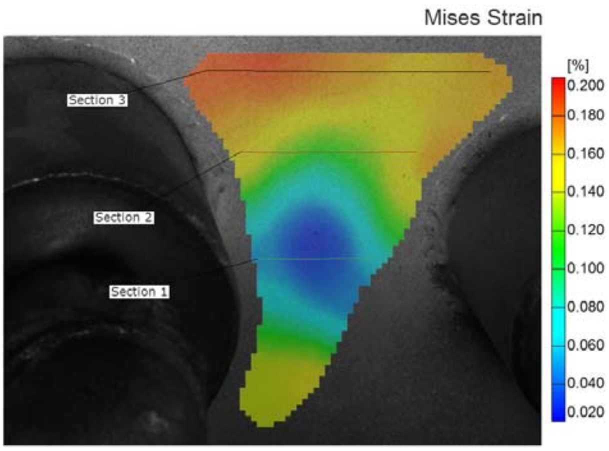


Figure 3. Von Mises strain distribution between nozzles for internal pressure 1.5 MPa.

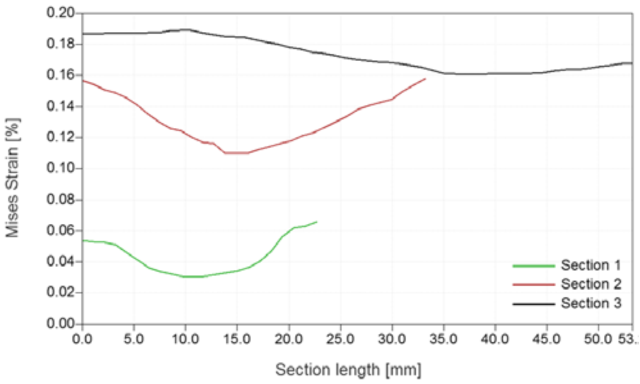


Figure 4. Diagram of Von Mises strain for characteristic sections.

FINITE ELEMENT ANALYSIS (NUMERICAL MODEL)

Static nonlinear finite element analysis of experimental model vessel is carried out. Finite element (FE) model of the vessel, including the metal support, is generated in the ANSYS (Fig. 5), and initial numerical simulations are performed in order to verify the model quality. The model used in the simulation had 601582 nodes and 300496 finite elements and is obtained by means of the iteration process which consisted of comparison between numerical results obtained for current, denser mesh, and values obtained in calculations with a coarser mesh. In each iteration process, the mesh step is refined until the difference in stress and strain values in two consecutive steps was less than 5%. Material properties are defined in ANSYS (steel with Young's elasticity modulus of 210000 MPa and Poisson's ratio of 0.3) and uniform pressure of 1.5 MPa.

From Figures 6 and 7 it can be seen that maximum Von Mises stress values are about 280 MPa, and strain values are about 0.23%. Based on the results obtained by experimental method and numerical simulation, the comparison of the maximal values of Von Mises strains is performed, and the difference between the two methods are up to 17.4%, so it is concluded that the FE model of the vessel with nozzles is well defined.

During the exploitation of a pressure vessel, /8/, bursting had occurred exactly at the location of the highest stress concentration, i.e. the connection between the nozzle and cylindrical shell, as shown in Fig. 8.

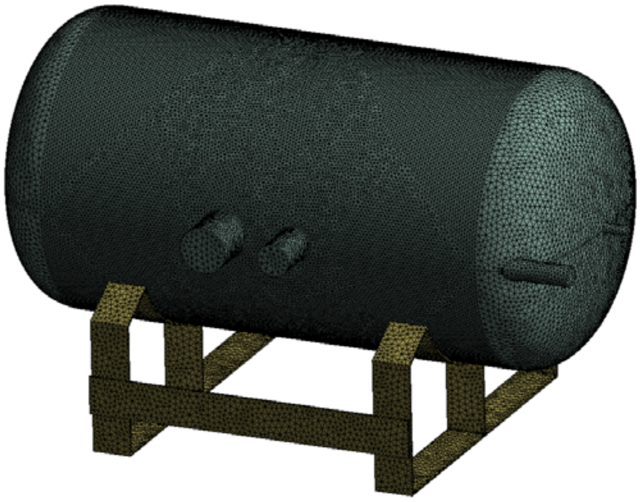


Figure 5. Finite element mesh for the model vessel.

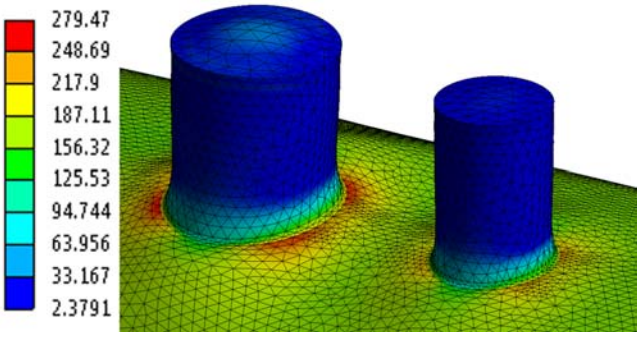


Figure 6. Von Mises stress distribution between nozzles.

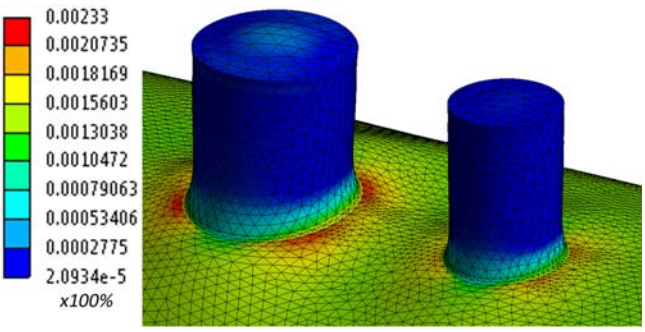


Figure 7. Strain distribution between nozzles.

Figure 8. Fractured surface of the test vessel.

EXTENDED FINITE ELEMENT METHOD (XFEM)

Cracks in structures can be tolerated in case they do not have significant influence on structural safety and do not cause noticeable economic effects. However, in cases when human safety and structural integrity are jeopardised, special attention needs to be paid to the problem of cracks. Modelling of cracks and development of methodologies for describing discontinuities around the crack have been the subject of numerous researches in the last decade, which lead to the development of entirely new calculation techniques. One of these is the Extended Finite Element Method (XFEM), which allows the representation of discontinuities independently from the finite element mesh.

The FEM has been used for decades for solving different engineering problems, but it has some restrictions in crack propagation simulations mainly because the finite element mesh needs to be updated after each propagation step in order to track the crack path. Extended Finite Element Method suppresses the need to mesh and remesh crack surfaces and is used for modelling different discontinuities in 1D, 2D and 3D domains. XFEM allows for discontinuities to be represented independently of the FE mesh by exploiting the Partition of Unity Finite Element Method (PUFEM), /13/. In this method, additional functions (commonly referred to as enrichment functions) can be added to the displacement approximation as long as the partition of unity is satisfied. The XFEM uses these enrichment functions as a tool to represent non-smooth behaviour of field variables.

There are many enrichment functions for a variety of problems in areas including cracks, dislocations, grain boundaries and phase interfaces. Recently, XFEM and its coupling with the level set method are intensively studied. The level set method allows for treatment of internal boundaries and interfaces without any explicit treatment of the interface geometry.

Due to the relatively short history of XFEM, commercial codes that have implemented the method are not prevalent. There are however, many attempts to incorporate the modelling of discontinuities independent of the FE mesh by either a plug-in or native support. Cenaero, /16/, has developed a crack growth prediction add-in Morfeo/Crack for ABAQUS which relies on the implementation of the XFEM method available in ABAQUS software (the functionality of ABAQUS is however limited to the calculation of stationary cracks). Problems such as static cracks in structures, evolving cracks and cracks emanating from voids, are numerically studied and the results are compared against analytical and experimental results to demonstrate the robustness of the XFEM and precision of Morfeo/Crack for ABAQUS, /17/.

Relying on the assumptions that the pressure vessel remains in the elastic regime everywhere, and that small-strain yielding conditions prevail in the vicinity of the crack front, the material is considered isotropic linear elastic and the simulation is carried out under the assumptions of linear elastic fracture mechanics (LEFM). An implicit representation of cracks is adopted in the spirit of the level set method.

Cracks are represented with the help of two signed distance functions that are discretized on the same mesh as the displacement field with first-order shape functions. The method for representing the cracks in this application is exactly the same as described in /18/. After each step of the propagation simulation, the stress intensity factors (SIFs) are computed from the numerical solution at several points along the crack fronts. Interaction integrals are used to extract the mixed-mode SIFs with the help of auxiliary fields.

Technique used here relies on a substructuring approach that decomposes the computation domain into several subdomains of two kinds: one or several safe subdomains, handled by the FEA code, and one or several cracked subdomains, handled by the XFEM code. The latter contains elements in the vicinity of initial cracks and in the region where they are approximately expected to propagate.

In case of the pressure vessel with two nozzles, it is decided to initiate a crack at the location of the highest stress concentration, and then monitor its growth through the structure. Finite element model with the crack is created in ABAQUS software and is shown in Fig. 9.

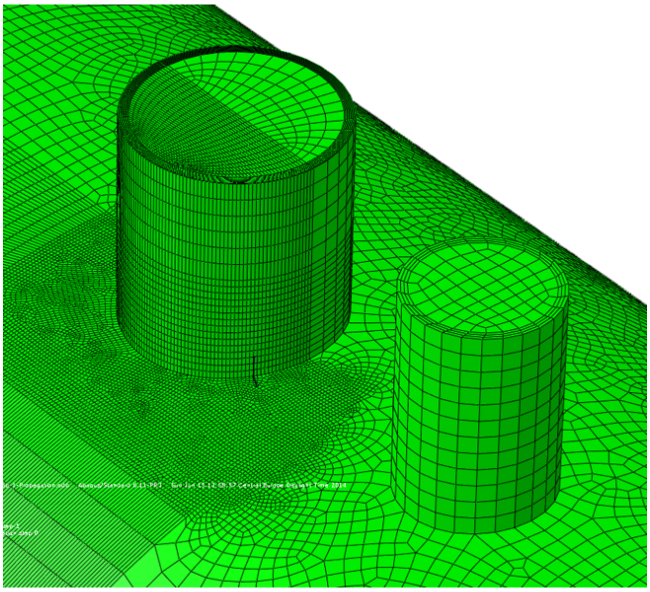


Figure 9. FEM model of critical area of maximum Von Mises stress between nozzles (ABAQUS software).

The defined mesh consists of hex finite elements, and it is refined around the crack at the area of maximum stress concentration (on one half of larger nozzle and along the wall of the vessel).

The first step in 3D analysis of crack propagation is its ‘opening’ (Fig. 10) and calculation of stress values in the pressure vessel. Stresses near the crack tip are represented by using three values, K_I, K_{II} and K_{III} – stress intensity factors for modes I, II and III.

In Figure 10, the equivalent Misses stresses around both tips of an open crack (one on the cylindrical shell, the other on the larger nozzle) are shown.

In Figure 12, the appearance of a crack in a cylindrical shell and in the nozzle is shown after 200 propagation steps.

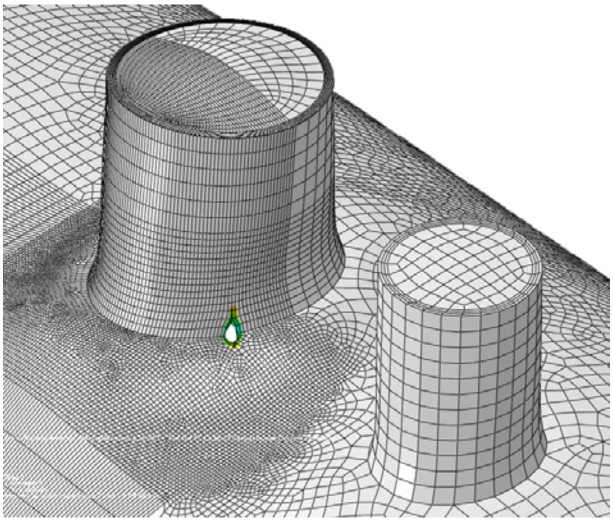


Figure 10. Crack opening

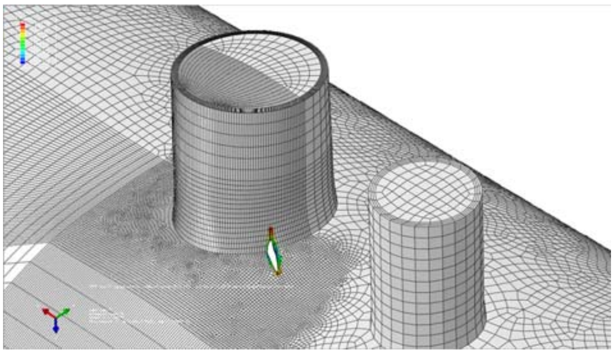
For the purpose of calculating stress intensity factors for modes I, II and III, the aforementioned Morfeo/Crack for ABAQUS is applied. It calculates stress intensity factor modes in nodes of the crack front and prepares a database of results, necessary for determining values in the next step of crack propagation.

Taking into account the direction of the propagation of a crack that has occurred (Fig. 9), forced plane crack propagation with a maximal propagation value 0.2 mm per step is selected. The number of propagation steps is 200.

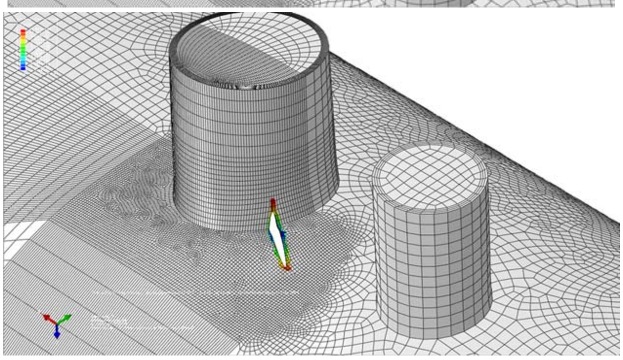
RESULTS

As it can be seen in Fig. 11, the crack moves along a vertical plane all the time, with two propagation fronts (one along the wall of the cylindrical part of the vessel, the other along the larger nozzle), wherein it moves through finite elements and ‘separates’ them, which is one of the main characteristics of XFEM.

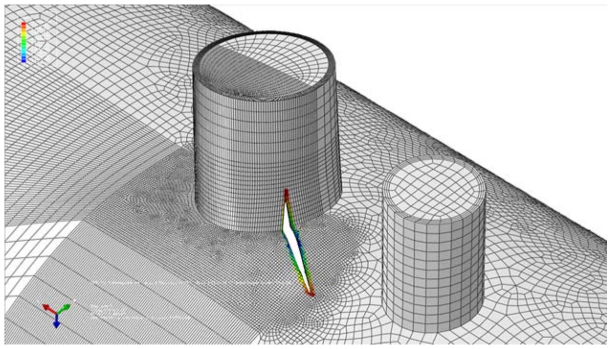
a)



b)



c)



d)

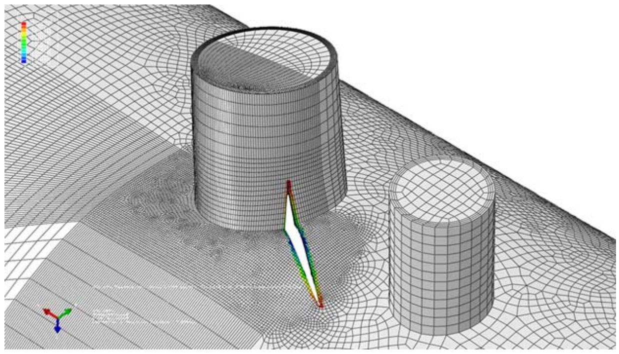


Figure 11. Stress distribution around crack tips after: a) 35; b) 78; c) 166; e) 196 steps of crack propagation.

Table 1 shows values calculated by Morfeo/Crack during each crack propagation step: curvilinear coordinate of each point along the crack front, coordinates of the crack front points in global *xyz* system, stress intensity factors for Modes I, II and III, as well as values of the equivalent stress intensity factor. The number of output values for each propagation step can be large and it depends on the number of points on the crack front, which again results from the density of the finite element mesh in propagation areas.

Figure 13 shows clear correlation between calculated SIFs values and crack length on the cylindrical shell (value of adjusted R squared is 0.994) and connection between two variables can be expressed in exponential form:

$$y = -7870.15 + 9507.90\, e^{0.00941x}$$

where y represents SIF, and x is the crack length.

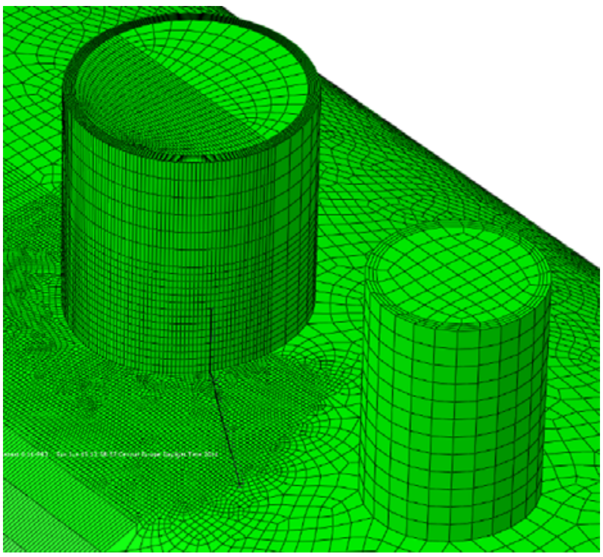


Figure 12. FEM model after 200 steps of crack propagation.

Table 1. Values calculated by Morfeo/Crack for ABAQUS for each step of crack propagation.

Curvilinear abscissa along the crack front	*x* co-ordinates of nodes on the crack front	*y* co-ordinates of nodes on the crack front	*z* co-ordinates of nodes on the crack front	Value of equivalent SIF	Value of SIF Mode I	Value of SIF Mode II	Value of SIF Mode III
0	-279.829	-20.0894	-188.029	2832.65	2721.71	-671.958	-471.578
0.126228	-279.819	-20.0605	-187.906	2833.04	2719.97	-667.67	-470.619
0.247086	-279.802	-20.0281	-187.791	2834.4	2713.78	-652.432	-476.212
0.375431	-279.782	-19.9928	-187.669	2838.67	2705.35	-648.453	-455.013
0.497137	-279.766	-19.9558	-187.555	2852.65	2681.32	-649.694	-414.534
0.62582	-279.752	-19.9153	-187.438	2866.21	2678.84	-651.252	-382.268

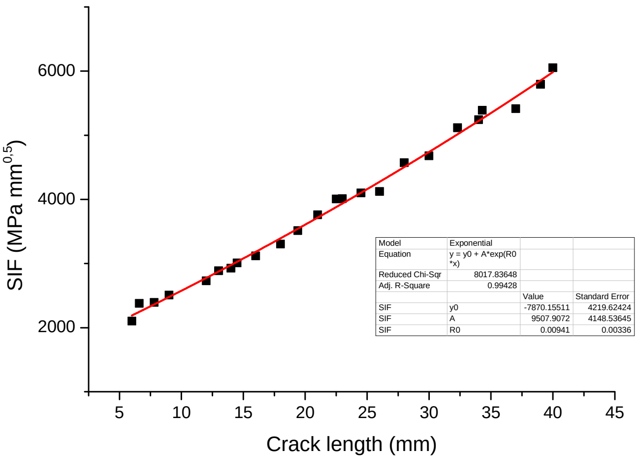


Figure 13. Variations of SIF values with crack length in the cylindrical shell.

A similar relationship can be obtained for the crack in the nozzle, too. Figure 14 shows the variations of cracks' lengths with steps of propagations.

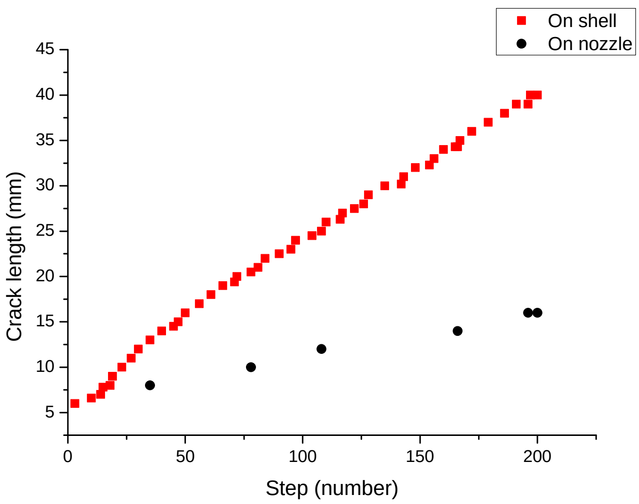


Figure 14. Variations of crack lengths in the cylindrical shell and in the nozzle with steps of crack propagations.

CONCLUSION

Based on the results presented here the following conclusions can be made:

- By applying 3D non-contact method for strain measurement, based on digital image correlation, the whole strain field in the vessel shell and around the nozzles is obtained. This experimental method does not depend on the type of material and geometry, thus it can be used to improve the knowledge about the behaviour of structures subjected to various types of loads.
- Static nonlinear finite element analysis of experimental model vessel is carried out. Finite element (FE) model of the vessel is generated in ANSYS. Results show very good agreement with experimental data.
- Once the numerical model of the vessel is verified, the initial crack is generated in the area with the largest strain concentration, since in practice, damages occur in almost the same area (Fig. 8). It can be claimed with a great deal of certainty that this is a critical area for a crack to appear in pressure vessels during their use.
- The crack is positioned in such a way that it reached both the cylindrical shell of the pressure vessel and the larger nozzle; therefore, during propagation it formed two fronts propagating simultaneously with a mutual angle of 90° (which corresponds to practical situations). Because of different wall thicknesses of the cylindrical shell and the larger nozzle, these two fronts cannot extend uniformly, which is also shown by the 3D simulation performed with the help of XFEM. Figures 12 and 14 show that the crack part in the larger nozzle is considerably shorter than the crack part in the cylindrical shell of the pressure vessel.
- Figure 13 shows that it is possible to establish correlation between the crack length, vessel wall thickness and stress intensity factor values in order to evaluate possible propagation speed of the crack in a certain area of the vessel. It is very important for pressure vessel integrity and it should be, by all means, included in future research in this area.

REFERENCES

1. Xue, M.D., Chen, W., Hwang, K.C., *Stresses at the intersection of two cylindrical shells*, Nuclear Engineering and Design, vol.154, 1995, pp.231-238.
2. Xue, M.D., Du, Q.H., Hwang, K.C., Xiang, Z.H., *An analytical method for cylindrical shells with nozzles due to internal pressure and external loads – Part 1: Theoretical foundation*, J of Pressure Vessel Technology, vol.132, 2010, pp.031206-1-9.
3. Xue, M.D., Du, Q.H., Hwang, K.C., Xiang, Z.H., *An analytical method for cylindrical shells with nozzles due to internal pressure and external loads – Part 2: Design method*, J of Pressure Vessel Technology, vol.132, 2010, pp.031207-1-8.
4. Nabhani, F., Ladokun, T., Askari, V., Reduction of Stresses in Cylindrical Pressure Vessels Using Finite Element Analysis, Finite Element Analysis – From Biomedical Applications to Industrial Developments, Dr. David Moratal (Ed.), 2012.
5. Skopinsky, V.N., *Numerical stress analysis of intersecting cylindrical shells*, J Pressure Vessel Technology, vol.115, 1993, pp.275-282.

6. Narale, P., Kachare, P.S., *Structural analysis of nozzle attachment on pressure vessel design*, Int. J Engineering Research and Applications, vol.2, No.4, 2012, pp.1353-1358.
7. Qadir, M., Redekop, D., *SCF analysis of pressurized vessel – nozzle intersection with wall thining damage*, Int. J Pressure Vessel and Piping, vol.86, pp.541-549, 2009.
8. Sang, Z.F., Xue, L.P., Lin, Y.J., Widera, G.E.O., *Limit and burst pressures for a cylindrical shell intersection with intermediate diameter ratio*, Int. J Pressure Vessel and Piping, vol. 79, 2002, pp.341-349.
9. Patel, D.M., Kumar, B., *Experimental method to analyse limit load in pressure vessel*, Int. J Modern Engineering Research, vol.4, no.10, 2014, pp.8-14.
10. Besnard, G., Leclerc, H., Hild, F., Roux, S., Swiergiel, N., *Analysis of image series through global digital image correlation*, J Strain Analysis for Engng. Design, vol.47, 2012, pp.214 -288.
11. Sedmak, A., Milosevic, M., Mitrovic, N., Petrovic, A., Maneski, T., *Digital image correlation in experimental mechanical analysis*, Structural Integrity and Life, vol.12, 2012, pp.39-42.
12. Balac, M., *Nozzles interaction influence on stress state and strains on cylindrical shell of the pressure vessel*, Ph.D. Thesis, University of Belgrade, Faculty of Mechanical Engng., 2014.
13. Babuska, I., Melenk, J.M., *The partition of unity method*, Int. J Numer. Meth. Eng., 40 (1998): 727-758.
14. Sukumar, N., Moës, N., Moran, B., Belytschko, T., *Extended finite element method for three – dimensional crack modelling*, Int. J Numerical Methods In Engng., (48), (2000): 1549-1570.
15. Liu, P.F., Zhang, B.J., Zheng, J.Y., *Finite element analysis of plastic collapse and crack behavior of steel pressure vessels and piping using XFEM*, J Failure Analysis and Prevention, 2012, vol.12, pp.707-718.
16. Cenaero Morfeo, available at http://www.cenaero.be/Page_Generale.asp?DocID=21686&la=1&langue=EN, last accesed on 3rd May 2015, (2015).
17. Grbović, A., *Investigation of fatigue life in superalloys structural components*, Ph.D. Thesis (in Serbian), University of Belgrade, Faculty of Mechanical Engng., Belgrade, Serbia, 2012.
18. Moës, N., Gravouil, A., Belytschko, T., *Non-planar 3D crack growth by the extended finite element and level sets — Part I: Mechanical model*, Int. J Numerical Methods in Engng., vol. 53, 2002, pp.2549–2568.