PARAMETRIC ANALYSIS FOR DETERMINATION OF FRACTURE MECHANICS PARAMETERS NEEDED FOR STRUCTURAL INTEGRITY ASSESSMENT OF PRESSURE VESSEL

Pejo Konjatić*, Ana Konjatić**, Filip Šakić*
*Mechanical Engineering Faculty in Slavonski Brod, University of Osijek, Slavonski Brod, Croatia, pekon@sfsb.hr, filip.sakic@sfsb.hr
**Industrial and Trade School, Slavonski Brod, Croatia, ana.konjatic@gmail.com

Keywords: parametric analysis, fracture mechanics parameters, structural integrity, pressure vessel, finite element method

ABSTRACT:

This paper describes approach to parametric study of influence of geometric parameters of one pressure vessel to fracture mechanics parameters needed for structural integrity assessment, which is commonly performed in order to assess safety of operation of pressure vessel when certain damage of pressure vessel is present. Cylindrical pressure vessel is subjected to investigation and in critical place (nozzle connection to pressure vessel), where maximum stress occurs, is used to introduce damage to pressure vessel in form of crack in order to conclude how it affects the safe operation of pressure vessel. Geometrical parameters of cylindrical pressure vessel which have most influence to stress concentration factor are varied as well as geometrical parameters related to fracture parameters, primarily damage size in context of crack length and depth. Certain analytical solution of fracture mechanics parameters and parameters needed for structural integrity assessment exist but for relatively simple geometries. In this case for obtaining results a finite element analysis is used, since more complex geometries can by analysed. 3D parametric model for analysis is prepared in 3D CAD software and parametric finite element model is prepared in software for analysis using finite element method. Results of conducted finite element analyses are presented in form of diagrams and tables and how individual geometrical parameters of geometry and damage influence safe operation of pressure vessel is explained.

1. INTRODUCTION

Nowadays pressure vessels are widely used for various applications in industrial and in private sector. If failure of pressure vessel occurs it is mainly caused due to pressure difference on interior and exterior surfaces of vessel. If such failure occurs, the catastrophic consequences can be expected depending on the size of pressure vessel, pressure difference, temperature and type of media contained inside pressure vessel. During the history of development and exploitation of pressure vessels, many accidents have happened. In order to prevent such dangerous situations and to ensure safe and reliable operation of pressure vessels, simplifying and reducing time needed for production at the same time, necessity for development of codes and standards for design and production of pressure vessels emerged [1] [2]. Like in any structural component, during production or exploitation of pressure vessels a certain damage or flaw can occur. Such flaw or damage can lead to failure of pressure vessel. Therefore, it is very important to detect flaws in order to repair pressure vessel or to predict, based on load, material and geometry of pressure vessel and geometry of damage, is operation of damaged pressure vessel safe. Recently structural integrity assessment procedures are developed for checking safe operation of damaged structural components including SINTAP [3]. Such procedures are based on fracture mechanics and obtaining fracture mechanics parameters of damaged component is needed in order to assess the structural component. Certain analytical solution for fracture parameters and parameters needed for structural integrity assessment exist but only for relatively simple geometries. Therefore, for geometries that are more complex a different approach is needed. That approach includes modelling of structural component with 2D or 3D CAD model, depending of analysed model, preparing finite element model with damage using finite element method and running finite element analysis for obtaining needed fracture mechanics parameters for structural integrity assessment.

2. SUBJECT OF INVESTIGATION

Since critical place on pressure vessel is usually connection of pressure vessel to nozzle it was planned to investigate one set of cylindrical pressure vessels with nozzle connection where diameter of nozzle is varied between 200 and 300 mm keeping thickness of nozzle and cylindrical wall of pressure vessel to 8 mm. On the place where maximum stress is expected, confirmed in previous investigations [4], damage in form of sharp semi-elliptic crack was implemented. It is the place of welded connection of nozzle to cylindrical body of pressure vessel shown on the Figure 1.

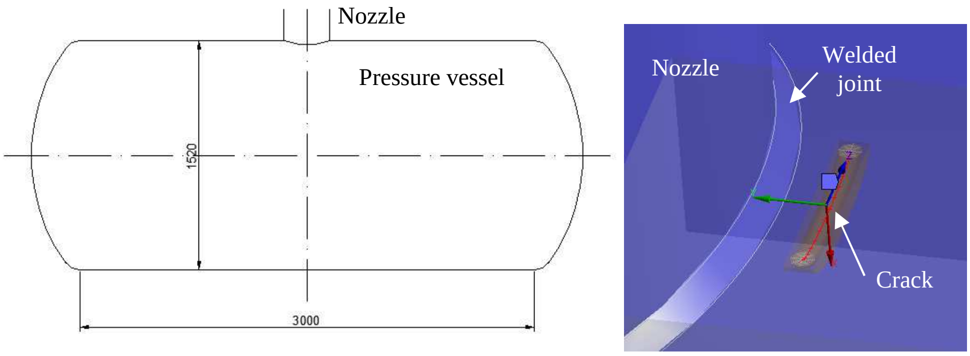

Figure 1: Geometry of pressure vessel and detail of crack position in vicinity of welded joint

Three-dimensional model in Computer Aided Design software SolidWorks [5] is modelled (Figure 2) and advantage of symmetry of geometry is used in order to model only one half of pressure vessel, since influence of stress and deformation distribution outside of vicinity of crack location can by neglected. Therefore, there is no mutual interaction of crack from other part of pressure vessel when symmetry boundary condition is applied. Different three-dimensional models are modelled using parametric modelling where changing variable of dimension of model, nozzle diameter, is set to parameter. Using parametric geometry saves time i.e. reduces time consuming process of modelling when multiple similar models are to be modelled with the same geometry but with different dimensions. Later, beside this parameter in finite element analysis, additional parameters related to crack geometry are added to already defined nozzle diameter parameter.

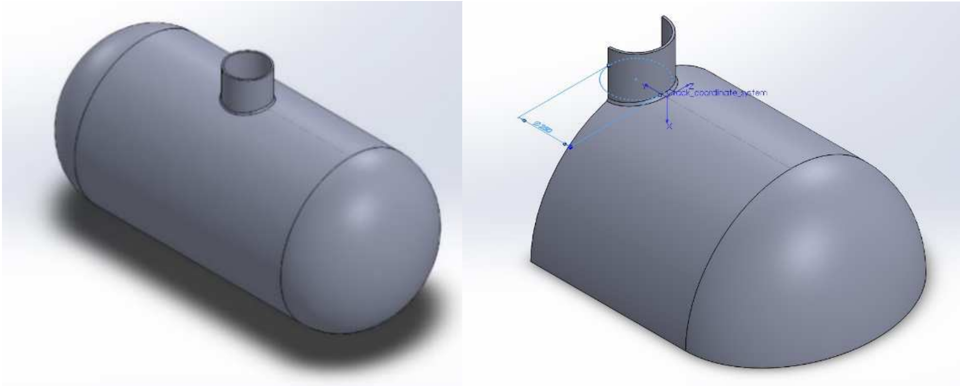

Figure 2: 3D model and one half of model used for geometry import to finite element analysis software

3. FINITE ELEMENT ANALYSIS

Three-dimensional finite element analyses were performed using ANSYS software [6] on parametric models created in SolidWorks 3D CAD modelling software. Additional parameters related to crack dimensions are defined. Crack length which was changed from 20 to 30 mm and crack depth ranged from 2 to 4 mm presenting maximal depth of damage to the half of the wall thickness of the pressure vessel. Nozzle diameter, crack length and crack depth were waried on three levels producing 27 different models for finite element method simulations. Numbers of elements and nodes in finite element mesh varied because of different dimension and largest model had 144033 nodes and 77146 elements. Finite element mesh was modified in such way that area of interest, around crack position, where higher stresses are expected had dense mesh in order to more precisely and accurately show stress distribution within wall of pressure vessel. On the other hand, less significant areas had very coarse mesh in order to reduce computation time needed for simulations. Crack front is modelled with singular elements needed do fully describe behaviour of material in front of the crack front and to determine stress intensity factor as most significant parameter in linear-elastic fracture mechanics which is needed for structural integrity assessment using SINTAP procedure. First row of singular elements along crack front was with diameter of 0,1 mm with 60 divisions along the crack front.

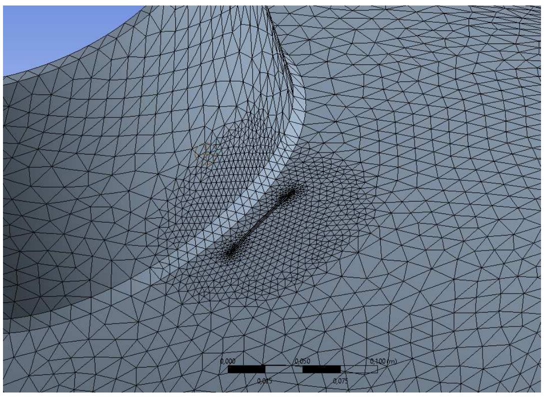

Figure 3: Finite element mesh of pressure vessel with detail on mesh around crack

In finite element analysis, performed to obtain stress intensity factor, it was needed to perform simulation with linear elastic material, therefore material of pressure vessel in analysis was considered as linear elastic with Young’s modulus of elasticity of 200 GPa and Poisson’s ratio of 0.3.

4. RESULTS OF FINITE ELEMENT ANALYSIS

For successful structural integrity assessment, using SINTAP procedure is required to determine fracture behaviour of pressure vessel. Fracture resistance of material, load pressure vessel are required as input parameters as well as damage geometry. In this investigation as output of finite element analysis stress intensity factor (SIF) is considered therefore as result of 27 performed finite element analysis dependence of SIF is presented in form of diagrams. SIF is determined for all three types of crack opening modes (for mode I - K_{I}, for mode II - K_{II} and for mode III - K_{III}) for crack position and loading in this investigation first mode is dominant (share of K_{II} and K_{III} is less than 4 % in effective value of SIF) and only K_{I} SIF in observed in analysis of results.

Three diagrams show results for SIF for three different nozzle diameters (Figure 4 for nozzle diameter of 200 mm, Figure 5 for 250 mm and Figure 6 for nozzle diameter of 300 mm for same thickness of pressure vessel wall thickness of 8 mm).

Generally, in all three cases it can be noted that SIF increases with crack depth and that crack length does not have large impact for shallow crack. That changes with increase of crack depth and combination of deeper crack with longer crack gives higher values of SIF. As for or shallow crack and for deepest cracks influence on SIF increase is stronger influenced by crack depth than by crack length. If comparing of all three diagrams is done it can be clear that SIF dependence on crack length and crack depth is similar for all three nozzle diameters, but larger diameter of nozzle connection gives lower values of SIF.

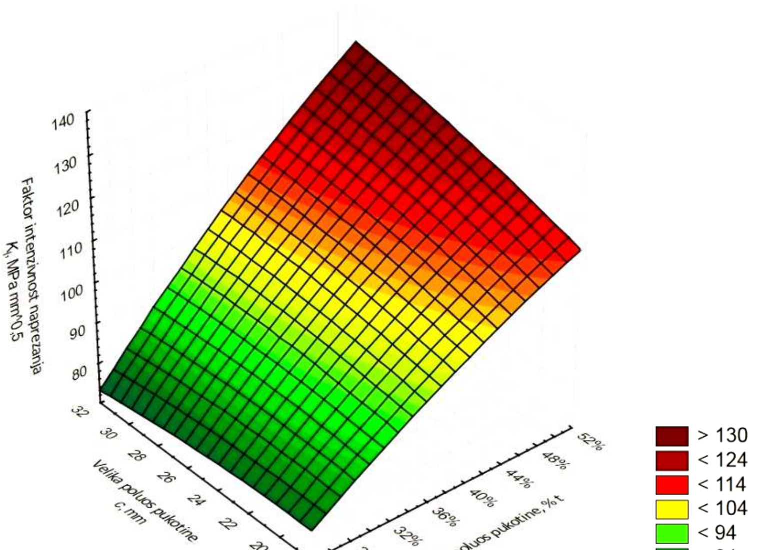

Figure 4: Stress intensity factor for pressure vessel shell thickness t=8 mm and nozzle diameter D=200 mm

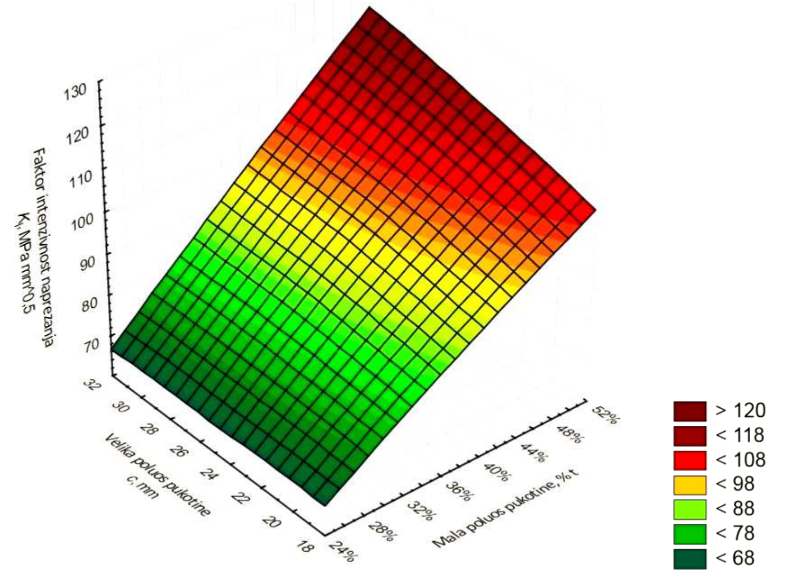

Figure 5: Stress intensity factor for pressure vessel shell thickness t=8 mm and nozzle diameter D=250 mm

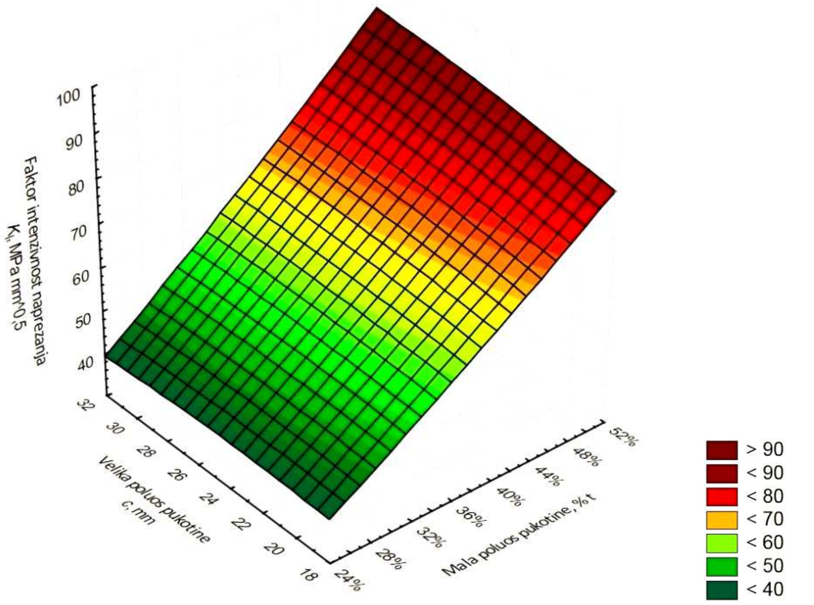

Figure 6: Stress intensity factor for pressure vessel shell thickness t=8 mm and nozzle diameter D=300 mm

5. CONCLUSION

Finite element analysis investigation of sets of cylindrical pressure vessels, where diameter of nozzle connection to pressure vessel was varied together with length and depth of crack in vicinity of welded nozzle connection, showed that dimension of nozzle diameter and crack dimension have influence on fracture mechanics parameter SIF. SIF is needed in order to successfully perform structural integrity assessment of damaged pressure vessel to determine safe operation of damaged pressure vessel. Nozzle diameter decrease has negative impact on SIF value causing SIF to increase. Crack length and crack depth also lead to SIF increase and increase of crack depth has more influence on SIF value increase. Therefore, it can be concluded that the most favourable situation for damaged pressure vessel give shallow crack, shorter crack and larger nozzle diameter and their impact on SIF is in such order as they are mentioned.

6. REFERENCES

[1] Konjatić P., Kozak D., Matejiček F., Ivandić Ž., Ergić T., Sertić, J.: Structural integrity assessment of welded cylindrical tank with torispherical heads used for underground storage of petrol, RIM 2009, Bihać, BIH, 2009. pp. 27-33
[2] Kozak D., Samardžić I., Hloch S., Konjatić P.: Overloading effect on the carrying capacity of cylindrical tank with torispherical heads for the underground storage of petrol, Manufacturing engineering - Vyrobne inžinierstvo, 2 (2008), VII, pp. 97-99
[3] SINTAP procedure, Final version: November 1999.
[4] Konjatić P., Šakić F., Kozak D., Beňo P.: Influence of geometry of pressure vessel nozzle connection on stress concentration factor, TEAM 2014, 2014. pp. 483-487
[5] SolidWorks v. 2013, Dassault Systèmes SOLIDWORKS Corp. 2013.
[6] ANSYS v.14., ANSYS Inc. 2013.
[7] Statistica v12., StatSoft, 2013.