Multiparameter Structural Optimization of Pressure Vessel with Two Nozzles

Martina Balac(✉) and Aleksandar Grbovic

Faculty of Mechanical Engineering, University of Belgrade,
Kraljice Marije 16, Belgrade, Serbia
mbalac@mas.bg.ac.rs

Abstract. Structural analysis of pressure equipment (vessels) has always been a huge challenge for researchers. Pressure vessels are usually subjected to different loads in exploitation and small defects can lead to failure of the equipment, which may result in loss of life, health hazards and damage of property. Modern approach of stress and strain analysis of the influence of welded nozzles on pressure vessels involves numerical and experimental testing. In this research, 3D Digital Image Correlation (DIC) method for analyzing full field surface strain and stress, including camera system in combination with Aramis software, was used. After determination of critical areas with highest von Mises stresses and strain concentrations, numerical analysis of equivalent 3D model was performed in Ansys Workbench software. The aim of this paper is to present detailed parameter optimization of pressure vessel with two nozzles based on finite element analysis (FEA) of the structure. Several geometrical parameters were varied to obtain the optimum geometry of the pressure vessel, capable of withstanding the service load without plastic deformation. It is shown that carried out optimization gives the minimum weight of pressure vessel with optimized wall and nozzle thicknesses for the given load.

Keywords: Pressure vessel · Digital Image Correlation method
Finite element analysis · Optimization · Response surface method

1 Introduction

Pressure equipment is the most widely used equipment in various industrial sectors and is usually subjected to different loads in exploitation. Many researches deal with stress and strain analysis of pressure equipment using both experimental and numerical methods [1–7]. Due to different technical and technological requirements, nozzles are often welded on vessel's shell producing geometrical discontinuity, which results in stress and strain increase in that area [2, 6, 7].

Regarding the appropriate structural design of the pressure vessel, the optimization approach attracted many researchers, both in practice and research [8–11]. Optimization of the location and size of opening (hole) in a pressure vessel cylinder was presented in the research of Hyder and Atif [12]. They analysed three thick-walled cylinders with internal diameter (20, 25 and 30 cm), having 30 cm height and wall thickness of 20 mm. Firstly, they conducted the analysis of pressure vessel cylinder

N. Mitrovic et al. (Eds.): CNNTech 2018, LNNS 54, pp. 148–158, 2019.
https://doi.org/10.1007/978-3-319-99620-2_12

without hole, calculating tangential, longitudinal, radial, and Von Misses stresses on cylinder. Secondly, the optimization of the hole size was carried out by drilling holes with diameters 4, 8, 10, 12, 14, 16 and 20 mm at the centres of all three thick cylinders. A method that directly minimizes the weight of the vessel by optimally determining the wall thickness is presented by Widiharso et al. [13].

Zhang and Yang showed in their paper [14] that modern optimization design combines mathematical programming with computer technology. They selected two geometrical parameters as design variables and the maximum volume of the pressure vessel as objective function. They found that different tolerances of design variables and constraints have effect on final design results. However, the design variables and the objective function are close to the optimal value along with the increase of iteration.

Bochare [15] showed that an appropriate location and size of the opening in a pressure vessel results in minimizing the stresses induced due to the stress concentration resulting from the end flanges and other attachments. The main aim of his work was to design and optimize the spherical and elliptical head profile with hole on the head. Analysis was done for elliptical and spherical heads without and with holes and by changing the diameters and distances between the holes.

Structural analysis of pressure vessel with and without reinforcement of nozzle for offset of 0, 8, 16, 24 and 32 inches from vertical centre line at central cross section with different inclination angles 0°, 15°, 30° and 45° was presented in [16]. Analytical and finite element analysis (FEA) of cylindrical pressure vessel with different vessel's head (hemispherical, standard ellipsoidal, dished shape and torispherical) has been presented in [17, 18]. Stress distribution in the pressure vessel was compared for diverse types of pressure vessel heads and it was concluded that stresses and deformation in hemispherical heads are the lowest.

Considering previous research papers, the aim of this research was to optimize several parameters that have high impact on stress and strain state of the pressure vessel with two nozzles.

2 Experimental Analysis

2.1 Basic Model

Experiments were performed on horizontal pressure vessel with the following dimensions: diameter of the vessel D = 378.4 mm, thickness of the vessels shell e_a = 1.5 mm, length of the vessel L = 770 mm. Pressure vessel was made of X5CrNi1810 material (EN 10088). On the cylindrical shell of the vessel, two nozzles – nozzle 1 DN 50 (Ø60.3 × 2.9 mm) and nozzle 2 DN32 DN 32 (Ø42.4 × 2.6 mm) were placed according to the appropriate EN standard. Nozzles were welded on the vessel and placed at an angle of 90° with respect to the longitudinal axis of the vessel. The nozzle's height influence on shell stress/strain state is negligible. Nozzles are placed at the minimum allowed distance, 98.3 mm, calculated according to the standard EN 13345-3 [6]. Experiment using Digital Image Correlation (DIC) method was conducted. This method [19–21] overcomes the limitations of conventional experimental methods and enables full-field displacement and strain measurement. At the beginning of 80's Peters and Ranson [22] first employed DIC. They assumed that there is a correlation in the intensity patterns of surface images taken before and after deformation. In the last decade, DIC technique has attracted full attention. DIC does not depend on the type of material or on the shape of the tested object and it has a wide range of applications, such as monitoring the strain and displacement in simple specimen [23, 24], or in the more complex structures [4, 25]. In this research, ARAMIS system developed by GOM has been successfully used to measure full strain field on the pressure vessel with two nozzles.

Equipment for experimental analysis consisted of previously mentioned optical system for 3D strain analysis (special sets of stereo cameras and lenses), software package Aramis, astand that allows the security and stability of sensors, devices to control the supply and image capture, PC systems and additional LED lighting (Fig. 1).

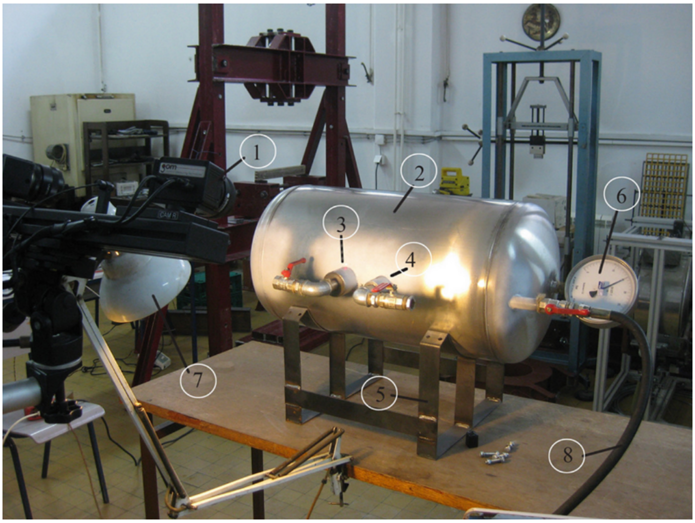

Fig. 1. Experimental setup, 1 - Stereo cameras; 2 - Vessel; 3 - Nozzle 1, DN50; 4 - Nozzle 2, DN32; 5 - Metal support; 6 - Pressure gauge; 7 - Illumination; 8 - Water pump connected to the pressure vessel

Areas of interest were defined to include the field between two nozzles and weld joint at the vessel's head. Parameters for basic Aramis system setup were: measuring volume 100 × 75, measuring distance 800 mm, camera angle 26°, calibration object CP20 90 × 72. Before starting the experiment, vessel's surface was prepared according to the tables in the instruction manual [26]. Several internal pressure values (0.5 MPa, 1 MPa and 1.5 MPa) were used in experiments.

2.2 Experimental Findings

The highest strain values on the pressure vessels obtained in experiments with the pressure 1.5 MPa are shown in Fig. 2.

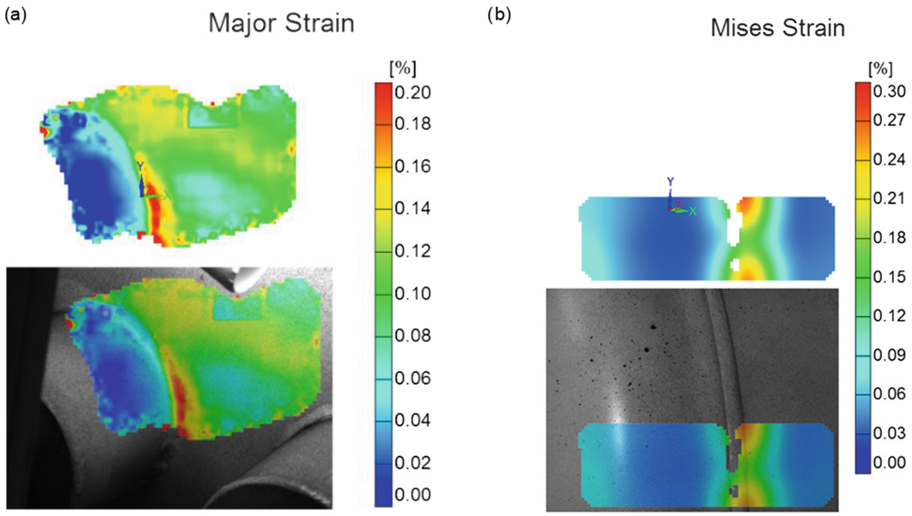

Fig. 2. Values of strain on pressure vessel for internal pressure of 1.5 MPa, (a) Strain distribution between the nozzles, (b) Strain on welded joint

Mises strain value near the nozzle 1 is 0.20% (Fig. 2a) and this area – where nozzle 1 is welded to the cylindrical surface – is the critical from the point of view of the crack initiation. Crack occurrence in this zone may lead to a complete failure of the vessel, as shown in [6, 27]. Figure 2b displays the strain distribution along the part of the weld joint of the vessel head. Value of the strain in that area varies from 0.10 to 0.25%, with dominant value of approximately 0.15%.

3 Finite Element Analysis

Based on the dimensions of the pressure vessel used in experiments, geometry of the vessel needed for numerical simulation was made in CATIA v5 software. Due to the symmetry of the vessel, only one quarter was designed and exported to Ansys Workbench software (Fig. 3). Applied boundary conditions and loads matched those used in experiments, while several different mesh densities had been used until the most appropriate was obtained. Final mesh consisted of 145,701 nodes and 72,732 solid elements. After applying pressure load, static nonlinear FEAs were carried out using material data obtained previously in the experiment with specimen [28]. The main purpose of these numerical simulations was verification of the developed model and – as it can be seen in Fig. 4 – strain values obtained in simulation were close to those obtained in the experiment. The highest strain values on outer surface appeared near nozzle 1 (Fig. 4a), in the same area where it appeared in experiment, with the values between 0.18% and 0.22% (max. value in experiment was 0.20%). Maximum strain in simulation 0.40% was observed on inner surface of a cylinder near nozzle 1 (Fig. 4b); however, this value couldn't be verified because no sensors have been used inside the pressure vessel. Nevertheless, values of strain measured on visible surfaces

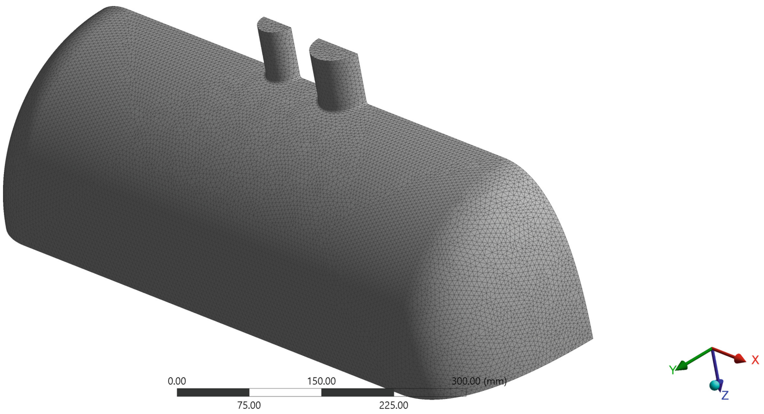

Fig. 3. Model of 1/4 of the vessel and finite element mesh used in FEA.

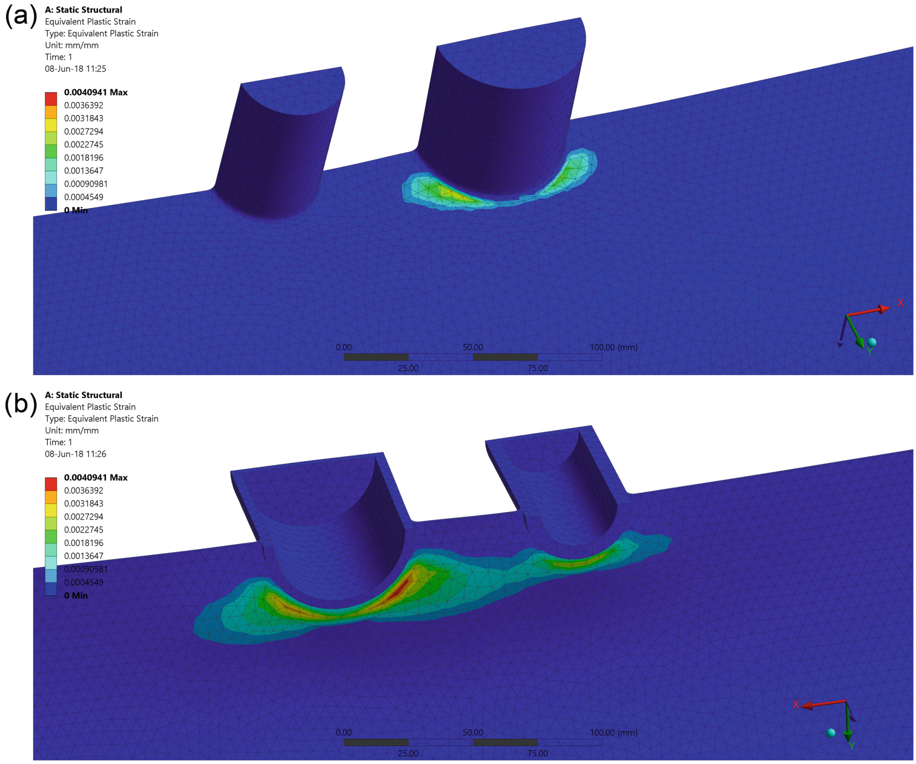

Fig. 4. Equivalent strain distribution on vessel for internal pressure of 1.5 MPa, (a) Strain near nozzle 1 is between 0.18 and 0.22%, (b) Max strain on inner surface

proved that numerical model was well defined. (It is worth mentioning that strain values near head's weld joint were about 0.15% and that maximum equivalent stress on vessel was around 200 MPa).

3.1 Optimization of the Pressure Vessel Geometry

After numerical model verification, next step in a research was optimization of the pressure vessel geometry with an aim of reducing its weight. Optimization design is developed in recent years and has been adopted by many researchers because it helps them to find the best solution from numerous design schemes. It combines mathematical programming with computer technology. In optimization design several steps should be followed: (1) Formation of parametric FEA file that includes unit types, inputting real constants, material parameters, verified solid model, mesh, constraints, loading conditions and required results; (2) Building of corresponding parameters' variables; (3) Executive optimization calculation, and (4) Testing and selecting the optimized results. Researches should be aware that there will be a series of feasible and infeasible design schemes throughout the process of optimization and that they must choose the best scheme and test the rationality of its results.

In a case of pressure vessel with two nozzles, four parameters – supposed to be the most influential on strain distribution under service load – were chosen: wall thickness of nozzle 1, wall thickness of nozzle 2, thickness of the vessel head and thickness of the cylindrical surface (Fig. 5). Upper and lower boundaries for each parameter have been selected after parameter choice. At the end of optimization process, the best parameter value should have been within defined boundaries.

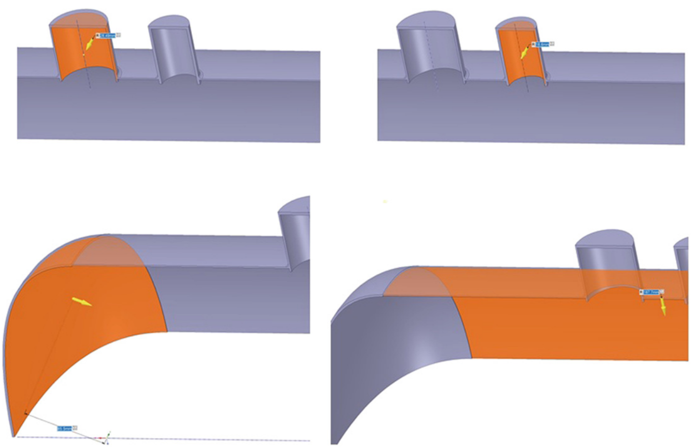

Fig. 5. Parameters optimized using Response Surface Optimization method

Next step in optimization, after parameters selection, was Design of Experiment (DOE). The purpose of a DOE was to gather representative set of data to compute RS and then run RSO. A set of design points was defined using different combinations of values for all four parameters within predefined boundaries. For that purpose, Central Composite Design (CCD) scheme was used. CCD is five-level fractional factorial design suitable for calibrating the quadratic response model [28]. There are 5 types of CCDs available in Ansys Workbench each with their own benefits and drawbacks: Auto Defined, Face Centred, Rotatable, VIF (Variance Inflation Factor) Optimality and G-Optimality. In optimization of pressure vessel Face Centred CCD was used.

It is important to emphasize that the RS accuracy greatly depends on the DOE scheme, and especially the number of design points that were calculated. In a DOE study, the amount of design points increases quickly as the number of input parameters increases, which can reduce efficiency of the analysis process. It is recommended to exclude unimportant input parameters from the DOE sampling in order to reduce time to solution. The parameter correlation tool allows researcher to identify unimportant parameters [29]. However, in pressure vessel optimization all four parameters were identified as important and 25 design points were generated. Calculation took about 120 min (I7 CPU at 3.5 GHz, 64 GB of RAM) and, as a result of good DOE, several response surfaces were obtained (Fig. 6).

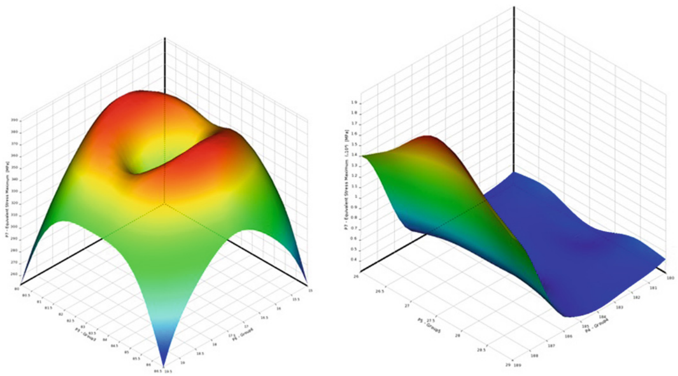

Fig. 6. 3D Response Surfaces representing relations between equivalent stress and different optimization parameters

Finally, the last step was to conduct RSO. One thousand screening samples had been generated first, followed by definition of objectives and constraints. Objective was to minimize the mass of the vessel, while the maximum value of equivalent stress 300 MPa was chosen as constraint. This was an arbitrary value, approximately 50% higher than 0.2% proof strength of the used steel. Idea was to allow higher stresses in pressure vessel to evaluate plastic strain, but still significantly lower than tensile strength; according to the materials database [30] tensile strength is between 500 and 750 MPa. After RSO had been done, table of optimization gave three “candidates” (i.e. combinations of parameters) based on specified goal and constraint. Trade off study was then conducted to reveal feasible points and, eventually, the best candidate was selected. Original model was updated with new values of parameters and numerical simulation was carried out once again. Values of strain in the area of interest on the optimized pressure vessel are given in Fig. 7.

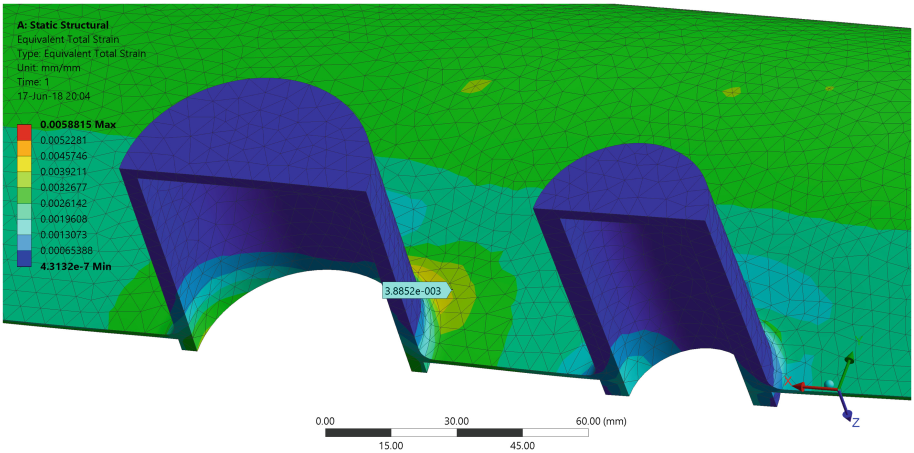

Fig. 7. Equivalent strain distribution on pressure vessel with optimized geometry

4 Discussion

Results obtained in experiment and FEA showed good agreement, i.e. both approaches indicated the same area of the highest strain in the vicinity of the one of the nozzles. This was the sign that all elements of the numerical model (geometry, load, boundary conditions) were well defined.

Response Surface Optimization (RSO) method was used to optimize geometry of the pressure vessel parts (shell, head and nozzles). Response surfaces (RS) are functions of different nature where the output parameters are described in terms of the input parameters. They provide the approximated values of output parameters everywhere in the analysed design space, without need to perform a complete solution. There are six RS types in Ansys Workbench, but in this research genetic aggregation (GA) type was used. GA runs an iterative genetic algorithm to find the best RS type and settings for each output parameter. It selects the best ones and combines them to build an "aggregation" of several RS. This results to the best RS quality and different settings for each output parameter. The goal is to meet 3 main criteria to obtain the best RS: accuracy, reliability and smoothness.

Even if the GA runs on several CPUs, this method can still be long when dealing with high number of design points and/or parameters (mesh density also influences the time to result). RSO methods are suitable for problems using up to 15 input parameters, but – due to the possibility of very long calculations – in this research 4 parameters were used as input and 2 as the output parameters. The objectives of research were to reduce the mass of the pressure vessel for given load and to keep the vessel below the predefined stress value.

5 Conclusion

The safety performance of pressure vessels is arousing increasing attention of many researches. In the past, they've been focusing on the stress and strain analysis and plastic deformation of pressure vessels. At present, the main aim is to solve the problem of shape optimization of the vessels in order to save materials and energy during manufacturing, but preserve reliability during exploitation. This can directly reduce costs and, consequently, the price of this equipment.

This paper presents a numerically based attempt in pressure vessel optimization. Objective of the optimization was minimization of the vessel's mass by varying the thicknesses of nozzles' walls as well as of vessel head and cylindrical surface. Using Response Surface Optimization method mass of the vessel was reduced from 3.642 to 3.383 kg, which gives the total reduction of 1.036 kg (considering that one fourth of the vessel was modelled). Thicknesses of nozzles' walls were increased by 0.85 mm (nozzle 1) and 3.2 mm (nozzle 2), while the thickness of the cylindrical surface was reduced by 0.3 mm (from 1.5 to 1.2 mm). Thickness of the vessel head remained almost the same: initial was 2.2 mm while final somewhat higher − 2.23 mm.

Figure 7 shows that strain value around nozzle 1 on optimized vessel is almost doubled (0.388% compared to 0.20% on original model and measured in experiment) which is expected considering the fact that thickness of cylindrical surface is now reduced, and the higher value of stress was allowed in optimization (new value of maximum equivalent stress is 315 MPa compared to 200 MPa on original vessel). Nevertheless, the main goal is achieved, and further investigations will show are these values of stress and strain acceptable (from the point of view of vessel integrity) and if they are, is there more room for further mass reduction. At least, additional methods of optimization can be tested and other parameters (like distance between the nozzles, for example) might be chosen.

Acknowledgements. Study presented in this paper is part of the Project TR 35031 financed by Ministry of Education, Science and Technological Development of Republic of Serbia.

References

1. 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. Press. Vessels Pip. **79**, 341–349 (2002). https://doi.org/10.1016/S0308-0161(02)00028-5
2. Petrovic, A., Balac, M., Jovovic, A., Dedic, A.: Oblique nozzle loaded by the torque moment: stress state in cylindrical shells on the pressure vessel. Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci. **226**(3), 567–575 (2012). https://doi.org/10.1177/0954406211415907
3. Bajic, D., Momcilovic, N., Maneski, T., Balac, M., Kozak, D., Culafic, S.: Numerical and experimental determination of stress concentration factor for a pipe branch model. Tehnickivjesnik Tech. Gaz. **24**(3), 687–692 (2017). https://doi.org/10.17559/tv-20151126222916
4. Mitrovic, N., Petrovic, A., Milosevic, M., Momcilovic, N., Miskovic, Z., Maneski, T., Popovic, P.: Experimental and numerical study of globe valve housing. Chem. Ind. **71**(3), 251–257 (2017). https://doi.org/10.2298/hemind160516035m

5. Maneski, T., Bajic, B., Momcilovic, N., Milosevic, M.V., Balac, M.: Determination of internal pressure value causing pipe branch model to plastically deform. FME Trans. **46**(2), 218–223 (2018). https://doi.org/10.5937/fmet1802218m
6. Balac, M., Grbovic, A., Petrovic, A., Popovic, V.: Fem analysis of pressure vessel with an investigation of crack growth on cylindrical surface. Eksploatacjai Niezawodnosc Maint. Reliab. **20**(3), 378–386 (2018). https://doi.org/10.17531/ein.2018.3.5
7. Lu, M.H., Yu, J.S., Chen, J.J.: The effect of analysis model on the stress intensity calculation for the nozzle attached to pressure vessel under internal pressure loading. Int. J. Press. Vessels Pip. **117–118**, 9–16 (2014). https://doi.org/10.1016/j.ijpvp.2013.10.004
8. Sathe, A.A., Maurya, V.R., Tamhane, S.V., Save, A.P., Nikam, P.V.: Design and analysis of pressure vessel components as per ASME Sec. VIII Div. III. Int. J. Eng. Dev. Res. **6**(1), 834–840 (2018)
9. Sahu, Y.K., Nagpal, S.: Design and optimization of a low pressure vessel. Res. J. Eng. **6**(7), 1–6 (2017)
10. Carbonari, R.C., Muñoz-Rojas, P.A., Andrade, E.Q., Paulino, G.H., Nishimoto, K., Silva, E.C.N.: Design of pressure vessels using shape optimization: an integrated approach. Int. J. Press. Vessels Pip. **88**, 198–212 (2011). https://doi.org/10.1016/j.ijpvp.2011.05.005
11. Kulkarni, A.A., Jatkar, K.H.: A review on optimization of finite element modelling for structural analysis of pressure vessel. Int. J. Eng. Trends Technol. (IJETT) **12**(1), 20–22 (2014)
12. Javed Hyder, M., Asif, M.: Optimization of location and size of opening in a pressure vessel cylinder using ANSYS. Eng. Fail. Anal. **15**, 1–19 (2008). https://doi.org/10.1016/j.engfailanal.2007.01.002. ISSN 2231-5381
13. Widiharso, H.S., Tauviqirrahman, M., Jamari, M.J.: Thickness optimization of pressure vessel for minimum weight using finite element method (FEM). Int. J. Eng. Technol. **8**(6), 2676–2682 (2017). https://doi.org/10.21817/ijet/2016/v8i6/160806225
14. Zhang, C., Yang, F.: Pressure vessel optimization design based on the finite element analysis. Appl. Mech. Mater. **65**, 281–284 (2011). https://doi.org/10.4028/www.scientific.net/AMM.65.281
15. Bochare, S.: Optimization of location and size of opening in a pressure vessel cylinder for spherical and elliptical head. Int. J. Emerg. Technol. **6**(2), 229–234 (2015)
16. Gupta, S.R., Desai, A., Vora, C.P.: Optimize nozzle location for minimization of stress in pressure vessel. Int. J. Adv. Eng. Res. Dev. (IJAERD) **1**(6), 1–14 (2014)
17. Digvijay, K., Jewargi, S.S.: Stress analysis of pressure vessel with different type of end connections by FEA. Int. J. Innov. Res. Sci. Eng. Technol. **4**(5), 2769–2775 (2015). https://doi.org/10.15680/IJIRSET.2015.0405016
18. Lawate, S., Deshmukh, B.B.: Analysis of heads of pressure vessel. Int. J. Innov. Res. Sci. Eng. Technol. **4**(2), 759–765 (2015). https://doi.org/10.15680/ijirset.2015.0402098
19. Orteu, J.J.: 3-D computer vision in experimental mechanics. Opt. Lasers Eng. **47**, 282–291 (2009). https://doi.org/10.1016/j.optlaseng.2007.11.009
20. Sedmak, A., Milosevic, M., Mitrovic, N., Petrovic, A., Maneski, T.: Digital image correlation in experimental mechanical analysis. Struct. Integr. Life **12**, 39–42 (2012)
21. Kawabayashi, H., Matsuda, H., Zhao, C., Konzuma, H., Yamashita, T.: Modeling cylindrical shell with initial imperfections by using optical method. In: The Fifth China: Japan Joint Seminar for the Graduate Students in Civil Engineering, Shanghai, China, pp. 181–186 (2008)
22. Peters, W.H., Ranson, W.F.: Digital imaging techniques in experimental stress analysis. Opt. Eng. **21**(3), 427–431 (1982). https://doi.org/10.1117/12.7972925

23. Rodrigues, L.D., Freire, J.L., Vieira, R.D., Castro, J.T.P.: Strain analysis of thin pipe pressure vessels using digital image correlation. J. Mech. Eng. Autom. **4**(2), 63–72 (2014). https://doi.org/10.5923/j.jmea.20140402.03
24. Chen, F., Chen, X., Xie, X., Feng, X., Yang, L.: Full-field 3D measurement using multi-camera digital image correlation system. Opt. Lasers Eng. **51**, 1044–1052 (2013). https://doi.org/10.1016/j.optlaseng.2013.03.001
25. Revilock, D.M., Thesken, J.C., Schmidt, T.E., Forsythe, B.S.: 3D digital image correlation of a composite overwrapped pressure vessel during hydrostatic pressure tests. In: Proceedings of the 2007 SEM Annual Conference and Exposition on Experimental and Applied Mechanics, Springfield, MA, pp. 1–15 (2007)
26. https://support.gom.com. Accessed 21 Nov 2016
27. Balać, M., Grbović, A., Petrović, A.: A numerical predictions of crack growth in a pressure vessel with welded nozzles. Struct. Integr. Life **15**(1), 55–61 (2015)
28. 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 Engineering (2014)
29. www.ansys.com. Accessed 15 Apr 2018
30. http://www.steelnumber.com/en/steel_composition_eu.php?name_id=100. Accessed 21 Mar 2018