NUMERIČKA ANALIZA NAPONSKOG STANJA POSUDE POD PRITISKOM

NUMERICAL ANALYSIS OF STRESS STATE OF PRESSURE VESSEL

Bahrija Čuturić [1]
Ermin Bajramović [1]
Fadil Islamović [1]
Dženana Gačo [1]

[1]University of Bihać
Faculty of Technical Engineering

Ključne riječi:
naponsko stanje, numerička anliza, posuda pod pritiskom

Keywords:
stress state, numerical analysis, pressure vessel

Paper received:
16. 02. 2024.
Paper accepted:
29. 03. 2024.

Stručni rad

REZIME
Posude pod pritiskom, koje su zbog nepovoljnih radnih uslova u toku eksploatacije izložene visokim naponima, moraju biti pravilno projektovane uzimajući u obzir sve proračunske faktore, kako bi bile sigurne za vrijeme njihovog predviđenog vijeka trajanja. Provjera stanja posude, odnosno analiza opterećenja posude u radnim i ispitnim uslovima, je obavezujuća. Numerička analiza u novije vrijeme predstavlja jednu od najznačajnijih metoda sagledavanja naponskih stanja svih konstrukcijskih elemenata, pa tako i posuda pod pritiskom. Omogućava precizno određivanje mjesta najvećih napona i deformacija. U okviru ovog rada urađena je analiza napona veritikalne cilindrične posude pod pritiskom tipa VS 110 sa spiralom za zagrijavanje i hlađenje, zapremine 3750 litara i radnog pritiska 3 bara, koja se koristi za pripremu lijekova u farmaceutskoj industriji. Proračun posude urađen je prema standardu EN 13445 i pravilnicima o posudama pod pritiskom. Zatim je urađena numerička analiza naprezanja u programskom paketu Autodesk Inventor 2023. Rezultati numeričke analize pokazuju da se najveći naponi javljaju u torisferičnim dijelovima danca. Nadalje, cilindrični priključci na plaštu, kao i na torisferičnom dancu, djeluju kao ojačanje posude, jer u njihovoj blizini dolazi do smanjenja napona. Dobijeni rezultati i njihova analiza omogućavaju sveobuhvatni pristup ocjeni sigurnosti posude u eksploataciji.

Professional paper

SUMMARY
Pressure vessels that are exposed to high stresses, due to unfavorable working conditions during exploitation, must be properly designed, taking into account all calculation factors, in order to be safe during their expected lifetime. Checking the condition of the vessel, or analyzing the load of the vessel in working and test conditions, is mandatory. In recent times, numerical analysis represents one of the most important methods of analyzing the stress states of all structural elements, including pressure vessels. It enables precise determination of the location of the greatest stresses and deformations. In this paper, a stress analysis of a vertical cylindrical pressure vessel type VS 110 with a spiral for heating and cooling, with a volume of 3750 liters and a working pressure of 3 bars, which is used for the preparation of medicines in the pharmaceutical industry, was performed. The calculation of the vessel was made according to the EN 13445 standard and regulations on pressure vessels. Then a numerical stress analysis was performed in the Autodesk Inventor 2023 software package. The results of the numerical analysis show that the highest stresses occur in the torispherical parts of the vessel end. Furthermore, the cylindrical nozzles on the shell as well as on the torispherical ends act as a reinforcement of the vessel because there is a stress reduction in their vicinity.

1. INTRODUCTION

When designing pressure vessels, different loads that can occur simultaneously must be considered, taking into account the probability of their simultaneous occurrence. Design for adequate strength must be based on methods of calculating the load due to pressure in the vessel, and on experimental, numerical or other methods. When testing and analysing the state of pressure vessels, changes in stress and deformation states occur very often, as a result of geometric irregularities near the junction of two shells of the same or different geometric shapes. Also, nozzles or openings that are used for different purposes (emptying and filling the vessel, for placing measuring instruments, for control and interventions on the vessels) create problems. They are placed mainly on the periphery of the vessel and thus disturb the symmetry of the vessel, and since the geometrical parameters of the nozzles and the shell vary, they cause geometrical discontinuities on the vessels. Due to various technical requirements, the continuity of the surface contour is disturbed, which causes an increase in stress in that place.

The importance of the pressure vessels design is confirmed by the fact that many authors are still researching and analysing these issues due to different and complex conceptual solutions of geometry in order to reduce stresses and deformations. Respecting the codes of ASME section VIII [1] and directives 2014/6/EU [2] used in the design of pressure vessels, nowadays researchers are increasingly using numerical methods for easier and faster design.

Authors Yahya et al. in their work, using Finite Element Analysis (FEA), according to the allowable pressures, determined the vessel wall thicknesses in order to achieve acceptable maximum stresses. Furthermore, the design of the pressure vessel is achieved according to a series of ASME codes and engineering standards in order to reach the allowable design limits [3].

Research by Hazizi K. and Ghaleeh M. highlighted the importance of designing pressure vessels in accordance with ASME codes to ensure safety and prevent hazards associated with improper design and manufacturing. Through analysis, the researchers found that changes in the structure of the pressure vessel were necessary to reduce the stress. They observed an inverse relationship between deformation and tank shell thickness, while the safety factor increased linearly with shell thickness. Analysis of the stress distribution shows the cylindrical shell of the vessel was exposed to the highest stresses, while the ends, nozzles and foot rest were exposed to less stress. Using the finite element method, potential stress points inside the pressure vessel were identified, enabling the necessary changes to increase its safety [4].

In order to solve the problem of pressure vessel stress for ammonia purification, authors Mubarak et al. in their work proposed the analysis steps using the finite element method. They used three programs for modelling (Autodesk Inventor Professional 2017, Autodesk Nastran In-CAD 2017 and Autodesk Nastran In-CAD). The results show the highest stress occurs in the area of the joint between the manhole and the casing. At the end, the theoretical validation of the entire model was carried out, and the results were within the permissible limits [5].

In the article, Romensky describes a numerical analysis of the actual work for unit connecting walls with the bottom of a vertical cylindrical tank of 50,000 m³. The main results of numerical research and the obtained picture of the state of stress and deformation of the unit connecting walls to the bottom of the vertical cylindrical tank for different types of finite element models are presented and analysed [6].

In recent years, most sectors are replacing conventional materials with aluminium matrix materials. Rohitkumar conducted various mechanistic studies in the research. For the same geometric parameters of the steel pressure vessel, it performs FE analysis of the structural steel pressure vessel and determines the stresses for the internal pressures. A comparison is made between FEM results and theoretical results for validation purposes. It shows the design is safe, because the stress value is within the range of yield strength [7].

The main components of pressure vessels, the performance of cylinders and elliptical ends affects the normal operation of pressure vessels. Currently, there is no single theoretical formula for the junction area between the elliptical end and the cylinder. Therefore, the authors Wang et al. take a standard elliptical end as the object of research. The results show the theoretical formula for calculating the stress in the discontinuous region between the elliptical end and the cylinder is valid. By comparing and analysing theoretical and experimental stress values, the accuracy and applicability of the formula for calculating theoretical stresses in the discontinuous area is verified [8].

In their work, the authors Pandey and Jain study the effect of different bracing designs on pressure vessels in terms of static and free vibration analysis [9].

Prasanth and Sachidananda conducted research on the design and analysis of pressure vessels using the finite element method. Principal stress theory and distortion energy theory were used to validate their design, and the calculated results were compared with those obtained from FEA software. The researchers concluded that the maximum principal stress obtained from the calculations agreed with the FEA results, confirming the safety of the pressure vessel design [10].

Mali et al. performed a pressure vessel analysis, with the aim of proving that multi-layered pressure vessels have the ability to withstand higher internal pressures compared to solid-walled vessels. The analysis of the pressure vessel included the examination of various materials. The researchers concluded that the maximum stress of the pressure vessel remained within the yield stress limit of the selected material [11].

Tălu and Ţălu performed a pressure vessel analysis to compare the effects of using a flat end versus a hemispherical end. The study considered different orientations and different numbers of carriers. The researchers concluded that the von Mises and normal stresses in the pressure vessel are similar for both flat and hemispherical ends. They observed the closing stress of a flat head is approximately twice that of a container with a hemispherical end [12].

Devaraju and Pazhanivel performed a stress analysis on pressure vessels, taking into account internal pressure, self-weight and fluid weight. The design of the pressure vessel was developed using manual calculations, and the calculated stress values were compared with those obtained from ANSYS V12 software. Research findings showed that the stress on the pressure vessel shell remained well below the allowable stress limit for the vessel material [13].

Kumar does the modeling using Autodesk Inventor Professional 2023 software, and finite element analysis is performed to find the places where the stress is most concentrated. The greatest stresses are at the point where the nozzle is attached. Stress calculations were carried out using the finite element method and a parametric model was created. Multiple tests were conducted to test different positions of the cylindrical nozzle on the pressure vessel, as well as different orientations of the connection rather than a centrally located radial hole [14].

The study by Shen and Tang provides valuable information on the application of the Finite Element Method (FEM) in buckling analysis. The analysis includes the influence of geometric parameters, shape imperfections and loading conditions to accurately determine the critical buckling loads. Consideration of buckling effects contributes to a comprehensive assessment of pressure vessel performance and reliability [15].

Everything points to the contribution of conducting numerical analyses using finite element methods offered by various software packages in order to find out the stress and deformation states of pressure vessels. This paper aims to show the influence of geometric discontinuities on the stress state of a vertical cylindrical pressure vessel. An analytical calculation will be done on a concrete example. Then, using the finite element method, we want to show the stress analysis of the vessel exposed to working and test conditions using the Autodesk Inventor Professional 2023 software.

2. MATERIALS AND METHODS

2.1 Subject of research

The subject of research in this paper is a cylindrical vertical pressure vessel type VS 110 with a spiral for cooling and heating the medium, operating pressure 3 bar, volume 3750 l. A cylindrical vertical pressure vessel consists of a cylindrical shell, an upper and lower torispherical cover, a spiral, a support and several different openings. The pressure vessel selected for this study is a vessel used to prepare medicines in the pharmaceutical industry. It is designed as a vertical cylindrical for use in a fixed place on the legs with a cylindrical shell with a diameter of 1600 mm, a length of the shell 1440 mm and a total height of 2717 mm. It consists of two torispherical cones and a spiral for heating and cooling the medium as shown in Figure 1.

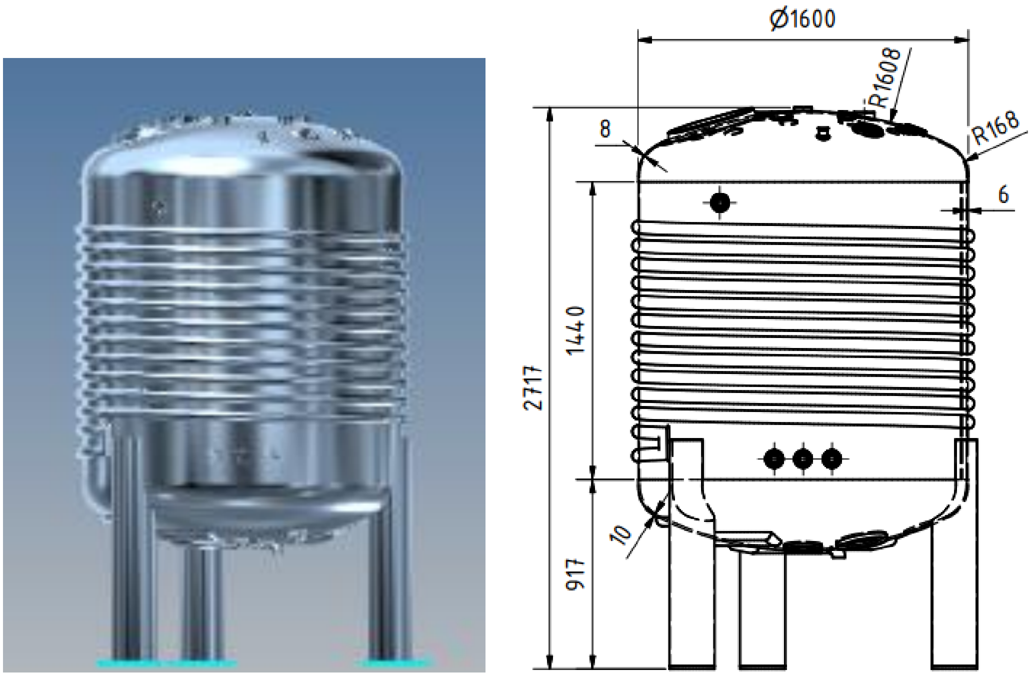

Figure 1 *The appearance and dimensions of the VS 110 pressure vessel*

For the pressure vessel, a preliminary design was made with the following parameters, which were taken as input data for designing and testing and are listed in Table 1 with a technical description.

Table 1 Technical description of pressure vessel VS 110

Description - technical characteristics	Values
Pressure vessel material	X2CrNiMo17-12-2 (1.4404)
Material properties	$R_{p0,2}$ = 270 MPa; $R_{p0,2/150\ °C}$ = 152 MPa; R_m = 500 MPa; $R_{m/150°C}$ = 410 MPa;
Young's modulus E	188970 MPa
Min./Max. pressure P	0/3 bar
Test pressure P	4,5 bar
The volume of the vessel V	3750 l
Mass of the vessel m	900 kg
Max. operating temperature T	150 °C
Vessel category	III

The total capacity of the vessel should work with a designed pressure of up to 3 bar for the required purpose.

2.2 Research methods

In this research, an analytical method will be used to analyse the condition of the vessel in question according to the EN 13445-3 calculation standard [16]. In the calculation part of the work, the condition of the internal pressure (working of 3 bar and test of 4.5 bar) will be taken into account. Given the pressure vessel in question has a complex geometry or load conditions, very often the calculation methods are inadequate for the accurate assessment of the vessel due to complex problems. Because of the complex problems, analysis using the finite element method (FEM) will be used to try to find a solution to the problem. In this way, by analysing the individual elements, and assuming the manner of their mutual connection, the entire problem is analysed. Regarding solving problems of continuum mechanics, FEM represents one of the most widespread modern numerical methods of approximate solving. FEM is used for designing and calculating all structures and elements, including pressure vessels. For these studies, FEM was performed using the Autodesk Inventor Professional 2023 software.

3. ANALYTICAL ANALYSIS OF STRESS

For the subject pressure vessel, a calculation was made for two cases of internal pressure loading, for a working pressure of 3 bar and a test pressure of 4.5 bar. The permissible nominal stress is calculated according to the EN 13445-3 standard. The dimensions of the vessel are: outer diameter De = 1590.5 mm, nominal thickness of the cylindrical shell e = 6 mm, thickness of the upper torispherical end es = 8 mm and the lower one es = 10 mm, allowance for corrosion C = 0 mm, permissible thinning of the walls δe = 0.38

3.1 **Stresses at working pressure p = 3 bar**

The maximum allowable stress of the vessel for the specified conditions:

$$f_d = \max\left[\left(\frac{\frac{R_{p1,0}}{T}}{1,5}\right);\min\left(\frac{\frac{R_{p1,0}}{T}}{1,2};\frac{\frac{R_{p1,0}}{T}}{3}\right)\right]$$

$$= \max\left[\left(\frac{181}{1,5}\right);\min\left(\frac{181}{1,2};\frac{410}{3}\right)\right]$$

$$= 136,7 \text{ MPa} \tag{1}$$

Stress in the cylindrical shell of the vessel:

$$f = \frac{P \cdot D_m}{2 \cdot e} = \frac{0,3 \cdot 1590,5}{2 \cdot 6} = 39,8 \text{ MPa} \tag{2}$$

Stress in the spherical part of the upper end of the vessel (R = 1600 mm):

$$f_s = \frac{P \cdot (R + 0,5 \cdot e_s)}{2 \cdot e_s} =$$

$$\frac{0,3 \cdot (1600 + 0,5 \cdot 8)}{2 \cdot 8} = 30,8 \text{ MPa} \tag{3}$$

Stress in the torus part of the upper end of the vessel (r = 160 mm):

$$f_y = \frac{P \cdot \beta \cdot (0,75 \cdot R + 0,2 \cdot D_i)}{e_s}$$

$$= \frac{0,3 \cdot 0,9(0,75 \cdot 1600 + 0,2 \cdot 1584)}{8}$$

$$= 51,2 \text{ MPa} \tag{4}$$

Considering that most of the openings are located on the torispherical end of the vessel, and as the standard requires, it is necessary to make an additional calculation for the openings. First of all, it was necessary to check whether the openings were isolated. An opening is considered isolated if the following condition is satisfied:

$$L_b \geq a_1 + a_2 + l_{so1} + l_{so2}$$
$$138 \geq 42,5 + 42,5 + 93,64 + 93,64$$
$$138 \geq 272,29 \tag{5}$$

If the condition of isolated openings is not satisfied, it means these are the adjacent openings. According to the equation, with input data of d_{eb} = 85 mm, $e_{a,b}$ = 14 mm and lb = 45 mm, the stress near the opening is:

$$f_s = P\left(\frac{Ap_{Ls}}{Af_{Ls}} + \frac{1}{2}\right) = 0,3\left(\frac{54631,4}{291,5} + \frac{1}{2}\right)$$

$$= 56,4 \text{ MPa} \tag{6}$$

3.2 **Stresses at test pressure p = 4,5 bar**

The maximum allowable stress of the vessel for the specified conditions:

$$f_{test} = max\left[\left(\frac{R_{p1,0/Ttest}}{1,05}\right);\left(\frac{R_{m/Test}}{2}\right)\right]$$

$$= max\left[\left(\frac{270}{1,05}\right);\left(\frac{530}{2}\right)\right]$$

$$= 265 \text{ Mpa} \tag{7}$$

Stress in the cylindrical shell of the vessel:

$$f = \frac{P \cdot D_m}{2 \cdot e} = \frac{0,45 \cdot 1590,5}{2 \cdot 6}$$

$$= 59,65 \text{ MPa} \tag{8}$$

Stress in the spherical part of the upper end of the vessel (R = 1600 mm):

$$f_s = \frac{P \cdot (R + 0,5 \cdot e_s)}{2 \cdot e_s}$$

$$= \frac{0,45 \cdot (1600 + 0,5 \cdot 8)}{2 \cdot 8} = 45,2 \text{ MPa} \tag{9}$$

Stress in the torus part of the upper end of the vessel (r = 160 mm):

$$f_y = \frac{P \cdot \beta \cdot (0,75 \cdot R + 0,2 \cdot D_i)}{e_s}$$

$$= \frac{0,45 \cdot 0,9(0,75 \cdot 1600 + 0,2 \cdot 1584)}{8}$$

$$= 76,8 \text{ MPa} \tag{10}$$

Nozzle stress on the cylindrical shell of the vessel for adjacent openings, with inputs $d_{eb} = 85$ mm, $e_{a,b} = 14$ mm and $lb = 45$ mm, the stress near the openings is:

$$f_s = P\left(\frac{Ap_{Ls}}{Af_{Ls}} + \frac{1}{2}\right) = 0,45\left(\frac{54631,4}{291,5} + \frac{1}{2}\right) = 84,56\ MPa \quad (11)$$

4. NUMERICAL ANALYSIS OF STRESS

The pressure vessel that is the subject of research in this paper is loaded with internal pressure. Numerical load analysis was done for two pressure values. Namely, one value was taken for the maximum calculated pressure of 3 bar, and one value for the test pressure of 4.5 bar.

4.1 Numerical analysis results for a pressure of 3 bar

For a pressure load of 3 bar, the overall results (Table 2) and the Von Misses stress of certain parts of the vessel are shown (Figure 2):

Table 2 Analysis results for 3 bar pressure loading

The Title	Min. values	Max. values
Volume	3750 l	
Mass	900 kg	
Von Mises Stress	0 MPa	62 MPa
Safety factor	0	15

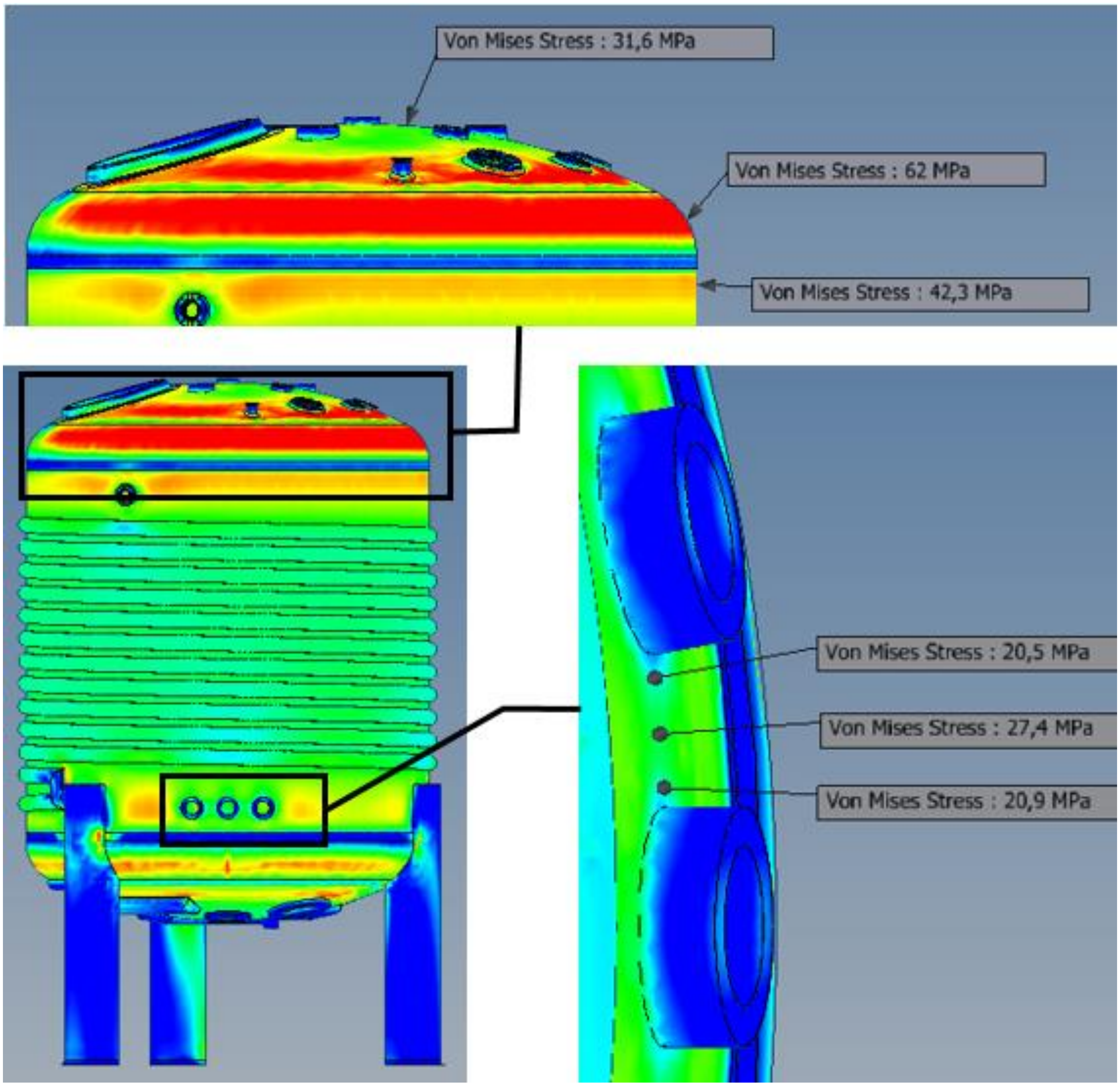

***Figure 2** Numerical analysis of Von Misses stresses for certain areas of the vessel*

4.2 Numerical analysis results for a pressure of 4,5 bar

For a pressure load of 4,5 bar, the overall results (Table 3) are shown. Figure 3 shows the Von Mises stress of the critical parts of the vessel. The red color shows the maximum stress while the blue color shows the area where the stress is 0 MPa.

Table 3 Analysis results for 3 bar pressure loading

The Title	Min. values	Max. values
Volume	3750 l	
Mass	900 kg	
Von Mises Stress	0 MPa	93,5 MPa
Safety factor	0	15

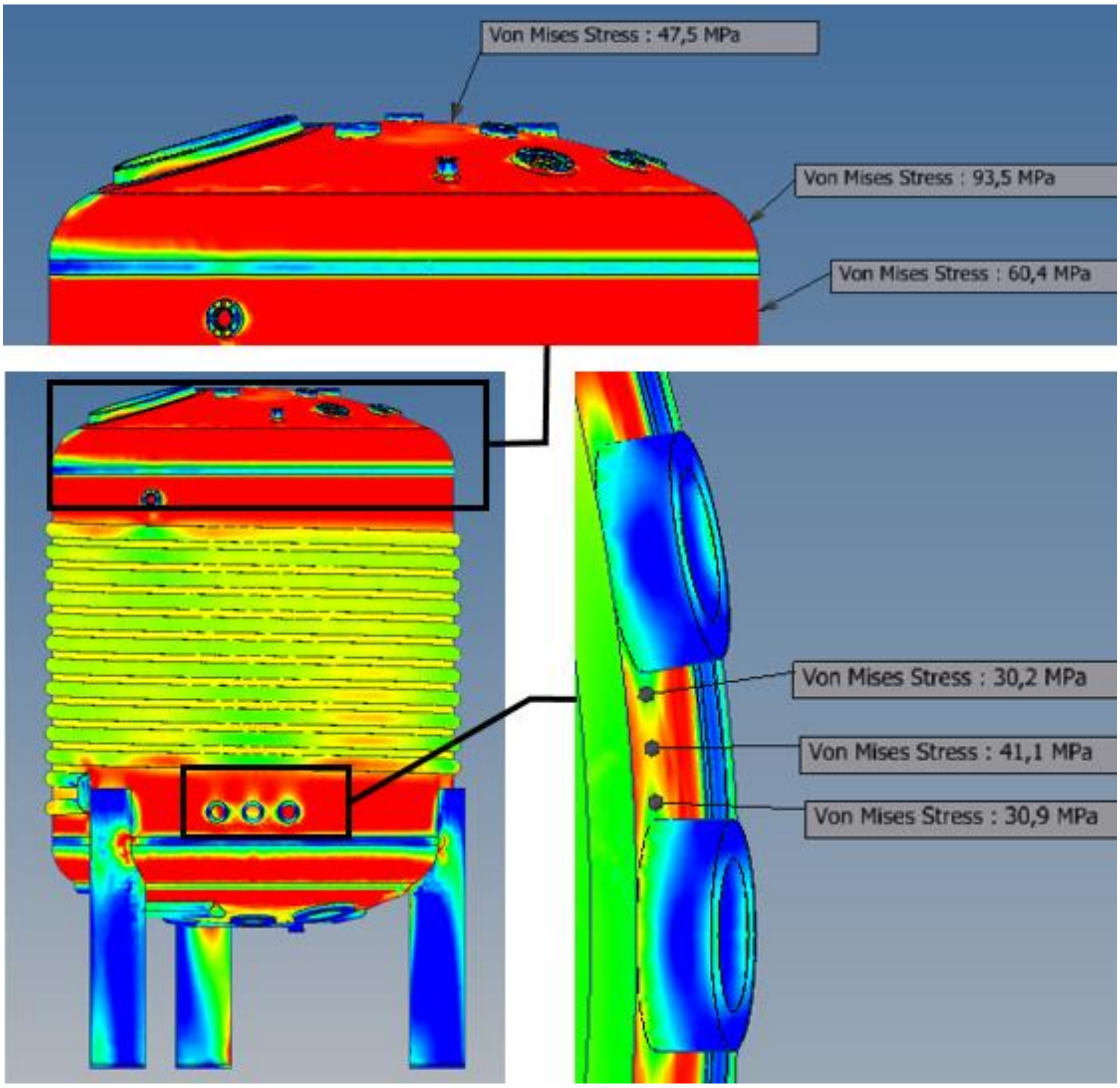

***Figure 3** Numerical analysis of Von Misses stresses for certain areas of the vessel*

5. ANALYSIS AND DISCUSSION OF RESULTS OF STRESS

As part of the finite element analysis, the Von Misses stress values were obtained for the entire pressure vessel, loaded with internal pressure. The highest values of the Misses stresses occur at the expected locations of the largest geometric discontinuities. The results of numerical simulations were verified by comparing the obtained values of Misses stresses with the stress values obtained by analytical analysis.

According to the certificate of the material used to make the vessel, it can be seen that the yield stress is $R_{p0,2}$ = 270 Mpa and the tensile stress R_m = 500 MPa, but the yield stress should also be taken into account, which at the working temperature of the vessel drops significantly and amounts to $R_{p0,2/150^\circ C}$ = 152 MPa, while the tensile stress at the working temperature of the vessel is $R_{m/150^\circ C}$ = 410 MPa.

For the internal pressure load value of 3 and 4.5 bar, a numerical analysis of the 3D vessel model was performed using the finite element method. It was shown that at this load the values of the Misses stresses are below the value of the permissible stress according to the EN 13445 standard.

According to the EN 13445 standard, the maximum allowed value of the nominal design stress is 136.70 MPa. The highest values of Von Misses stresses in the case of loading with an internal pressure of 3 bars occur in the torus part of the vessel end, and are around 62.7 Mpa, which represents the equivalent Von Misses stress. At the place of the connections, that is, near the opening, slightly higher values of Misses stresses occur on the outside of the nozzle, and between the nozzles, the values of Misses stresses are even lower by about 50% compared to the analytical values.

The maximum values of the analytical and numerical analyzes of the Misses stresses for different load locations with internal pressures of 3 bars are shown in Table 4.

Table 4 Analytical and numerical values of Misses stresses for loading with an internal working pressure of 3 bar

Load points Internal pressure P = 3 bar	Analytical by EN 13445	Numerical-Autodesk Inventor
	Von Misses stress MPa	Von Misses stress MPa
Cylindrical shell	39,8	42,3
Vessel end (R=1600 mm)	30,8	31,6
Vessel end (r=160 mm)	51,2	62
Near the nozzle	56,4	27,4

According to the EN 13445 standard, the maximum allowed value of the nominal design stress at the test pressure is 265 MPa. The maximum values of Von Misses stresses in the case of loading with an internal test pressure of 4.5 bar, also, occur in the torus part of the vessel end, and amount to about 93.7 MPa. The resulting stress represents the equivalent Von Misses stress and is lower than the nominal design stress. At the point of the nozzles and near the opening, slightly higher values of Misses stresses occur on the outside of the nozzles, while between the nozzles the values of Misses stresses are significantly lower and amount to about 40.9 MPa, which is even lower by about 50 % compared to the values obtained by analytical analysis. The maximum values of the analytical and numerical analysis of the Misses stresses for different load locations with internal pressures of 4.5 bar are shown in Table 5.

Table 5 Analytical and numerical values of Misses stresses for loading with an internal test pressure of 4,5 bar

Load points Internal pressure P = 4,5 bar	Analytical by EN 13445	Numerical-Autodesk Inventor
	Von Misses stress MPa	Von Misses stress MPa
Cylindrical shell	59,65	60,4
Vessel end (R=1600 mm)	45,2	47,5
Vessel end (r=160 mm)	76,8	93,5
Near the nozzle	84,56	41,1

6. CONCLUSION

In recent times, numerical analysis represents one of the most important methods of analyzing the stress states of all structural elements, including pressure vessels. Based on the research results, using the finite element method through the Autodesk Inventor Professional 2023 software, the stress state of a pressure vessel exposed to an internal working pressure of 3 bar and an internal test pressure of 4.5 bar is shown. The analysis included the cylindrical shell of the vessel, the torispherical end of the vessel and two nozzles on the cylindrical shell of the vessel. The analytical stress calculation was done according to the EN 13445-3 standard. Based on the analytical and numerical analyses, it is possible to conclude the following:

- By applying the finite element method, it was shown for both cases (working pressure 3 bar and test pressure 4.5 bar) that the highest values of Misses' stresses occur on the torus part of the end of the vessel.
- The stress values between nozzles and near them are not as present or large as the analytical calculation shows in this example.
- The minimum stress values occur on the cylindrical shell of the vessel. The spiral welded to the outer side of the jacket, which

serves to heat and cool the medium, acts as an additional reinforcement.

The application of numerical analyses significantly contributes to future research into the stress state of pressure vessels. Special attention is paid to the contribution of numerical analysis and its validity, and the possibility of application in the construction and calculation of pressure vessels, in order to reduce calculation time, increase the reliability of equipment, save material and energy for the manufacture of pressure vessels, resulting in lower costs of the finished product.

REFERENCES

[1] "2007 ASME Boiler and Pressure Vessel Code: An International Code." *American Society of Mechanical Engineers*, July 1, 2007.

[2] Safety and Health at Work EU-OSHA. "Directive 2014/68/EU - Pressure Equipment," May 6, 2023. https://osha.europa.eu/en/legislation/directive/directive-201468eu-pressure-equipment.

[3] Yahya, Najeeb A., Othman M. Daas, Nureddin O. Fahel Alboum, and Ahmed H. Khalile. "Design of Vertical Pressure Vessel Using ASME Codes." *The First Conference for Engineering Sciences and Technology*, November 30, 2018. https://doi.org/10.21467/proceedings.4.33.

[4] Hazizi, Kristaq, and Mohammad Ghaleeh. "Design and Analysis of a Typical Vertical Pressure Vessel Using ASME Code and FEA Technique." *Designs* 7, No. 3 (June 16, 2023). https://doi.org/10.3390/designs7030078.

[5] Arhami, Amir Zaki Mubarak, Andika Saputra, and Mohammed Ali. "Step by Step Stress Analysis of Ammonia Scrubber Vertical Pressure Vessel Using Finite Element Method." *IOP Conference Series: Materials Science and Engineering* 931, no1 (September 1, 2020). https://doi.org/10.1088/1757-899x/931/1/012002.

[6] Romensky, Denis. "Actual Work's Numerical Analysis for Support Joint of Vertical Cylindrical Tank." *E3S Web of Conferences* 402 (January 1, 2023). https://doi.org/10.1051/e3sconf/202340207025.

[7] Rohitkumar, S. Biradar. "Finite Element Modelling and Analysis of Pressure Vessel." *IJRASET*, no. 2321–9653 (September 14, 2022). https://doi.org/10.22214/ijraset.2022.46770.

[8] Wang, Zhanhui, Zhifang Zhang, Jinzhong Chen, and Jinrong Bai. "Stress Analysis and Applicability Analysis of the Elliptical Head." *Scientific Reports* 11, No. 1 (November 25, 2021). https://doi.org/10.1038/s41598-021-02397-7.

[9] Pandey, J., Jain, A. K. Performance Analysis of Pressure Vessel with various stiffener using FEA, International Journal of Science and Engineering Development Research, ISSN: 2455-2631, Vol.3, Issue 5, 481-494, May 2018, http://www.ijsdr.org/papers/IJSDR1805071.pdf

[10] Prasanth, Durga, and Hassan Sachidananda. "Design and analysis of pressure vessel." *International Journal of Mechanical and Production Engineering Research and Development* 9, No. 5 (October 6, 2019): 125-136

[11] Mali Amol, Hemant Bhosale, Dilpreet Singh Bedi, and Akash Modasara. "A Review Paper on Study of Pressure Vessel, Design and Analysis." *International Research Journal of Engineering and Technology (IRJET)* 04, No. 05, May 2017: 1369-1374

[12] Ţălu, Ştefan, and Ţălu Mihai. "Algorithm for Optimal Design of Pressurized Toroidal LPG Fuel Tanks With Constant Section", *Hidraulica*, June 27, 2018, 14–21.

[13] Devaraju, and Pazhanivel. "A Study on Stress Analysis for Design of Pressure Vessel." *International Journal of Mechanical and Production Engineering*

(IJMPE) 3, No. 11 (n.d.): 98–101.

[14] Kumar, Pavan, and Rambabu. "Investigation of Von Mises Stress and Stress Concentration Factor for Cylinders With Holes in Different Locations and Varying Pressure Using Analytical Method and FEA." *International Journal for Research in Applied Science and Engineering Technology (IJRASET)* 7, No. VII, July 2019.

[15] Shen, Jun, Yi Tang, and Y. H. Liu. "Buckling Analysis of Pressure Vessel Based on Finite Element Method." *Procedia Engineering* 130 (January 1, 2015): 355–63. https://doi.org/10.1016/j.proeng.2015.12.228.

[16] EN 13445-3: *Unfired pressure vessels - Part 3: Design*.

Corresponding author:
Bahrija Čuturić
University of Bihać
Faculty of Technical Engineering
Email:cuturic.b@hotmail.com
Phone: + 38762288576