Finite Element Analysis of Stress Distribution in Pressure Vessels under Internal Pressure

Umidjon Abdukodirov[*], *Rustambek* Urinov, *Muqaddam* Jurayeva, *Dildora* Mahkamova, *Makhbuba* Toshpulatova, *and Lolaxon* Teshabayeva

Department of Uzbek Language and Teaching Languages, Fergana State Technical University, Fergana, 150100, Uzbekistan

Abstract. This paper presents a finite element examination of the stress distribution in a cylindrical pressure-vessel segment under internal pressure based on the real simulation results received from the cloud-based platform SimScale. A three-dimensional linear static structural model was solved with the platform's integration of the Code Aster solver, employing an automatically generated unstructured tetrahedral mesh of approximately 1600 elements. A permanent support was put on one end of the vessel to prevent rigid-body motion. An internal pressure was supplied on the inside surface. Three pressure values of 1, 2 and 3 MPa were simulated. The corresponding peak von Mises stresses extracted from post-processing were 4.35 MPa, 8.70 MPa, and 13.06 MPa, respectively. The results demonstrate a strongly linear pressure–stress relationship (best-fit slope ≈ 4.355 MPa/MPa; intercept ≈ −0.007 MPa; $R^2 \approx 0.9999996$), consistent with linear elasticity and proportionate loading. Stress contours show that the maximum equivalent stress occurs at the inner wall, which is consistent with thick-cylinder elasticity where hoop stress is highest at the bore and drives the distortion-energy measure. Mesh-related influences, boundary-condition-induced gradients near the constrained end, and the limitations of linear modeling are critically assessed, and extensions to nonlinear plasticity and burst-pressure prediction are proposed.

Keywords. Pressure vessels; finite element analysis; internal pressure; von Mises stress; stress distribution; mesh influence; linear elasticity; thick-walled cylinder.

1. Introduction

Safety-critical components containing fluids at pressures that vary from ambient, when not designed or fabricated properly may leak the fluid or catastrophically rupture. Current practice thus involves a blend of code-based design and numerical analysis to identify the critical stresses and limit their uncertainty, particularly when the geometry (such as openings, nozzles, manways, junctions) causes stress concentrations that cannot be necessarily represented by the simple closed-form rules [1].

[*] Corresponding author: abduqodirov1982@gmail.com

Stress formula to perform analysis is still a vital tool for preliminary sizing and numerical results interpretation. Thin-walled cylinders have relationships between pressure and hoop and axial membrane stresses that are valid for equilibrium, and are often used within their range of validity [2]. In cases where thickness is not small, however, stresses vary across the thickness, and thick-cylinder (Lamé-type) solutions prove that hoop stress is greatest at the inner radius [3]. This theoretical motivation leads to the following points which motivated the present work: (i) the proportionality of the stress to pressure under linear elasticity, and (ii) the localization of the peak stress at the inner surface [2].

In recent years, the growing use of data-driven approaches to improve analysis efficiency has been a major research focus on pressure vessels. Machine learning tools have been proven to greatly speed up the local stress evaluation and are still quite accurate [12]. The finite element analysis has been proven to be a good tool for predicting the burst strength of thin and thick walled pressure vessels under elastic-plastic conditions and parametric FEA techniques has been found to be a good tool to determine the behaviour of a structure when it is made of different geometries and materials [13],[14]. Furthermore, complicated analytical and numerical investigations have been conducted for the behavior of thick-walled cylinders under non-uniform pressure conditions and revealed the influence of the stress distribution and material gradient through the cylinder thickness [15],[16].

The impetus for finite element analysis (FEA) is even greater in the discontinuity regions. Local stress fields control fatigue and integrity at vessel–nozzle junctions and simplified rule-based methods can differ as a result of assumption and geometric scope restrictions, thus calling for high fidelity FEA. Fatigue and thinning-damage analyses also yield similar results: peak stresses tend to focus around intersections, and local wall loss modifies the stress concentration factors [5].

In this larger context, the present study's aim is to purposely capture a canonical cylindrical pressure-vessel segment under internal pressure and to demonstrate, using actual cloud-based simulation results, the linear scaling of pressure and stress with one another, and the maximum stress experienced across the inner surface of the cylinder, under a controlled scenario. This study is conducted using only real simulation data.

The contributions are as follows: (1)A clear, albeit realistic, cloud FEA process (geometry, boundary conditions, meshing, post-processing) for industrial use on SimScale; (2) a detailed explanation of the reasons for choosing cloud FEA, along with examples from industry; (3) a comparison of cloud FEA with traditional FEA; and (4) an example on SimScale of an actual cloud FEA project. (3) verification of the above linear relationship by three real runs; and (3) a physics-based explanation, based on thin and thick cylinder theory, that the equivalent stress will be maximum at the inner surface [6]; and (4) a critical discussion about the influence of the mesh in the boundary conditions and the possible artefacts and how these are expected to be handled in future nonlinear and burst analysis..

2. Mathematical Formulation

Classical theory provides reference stress measures for a long cylindrical vessel with the inner radius r_i, outer radius r_o and wall thickness t=r_o - r_i, under the action of internal pressure p.

2.1. Thin-walled cylinder relations (membrane approximation)

If the wall is thin enough compared with its radius, the membrane stresses are assumed to be uniform through thickness and the radial stress is assumed to be small when compared to the membrane stresses [2]. The stresses in the hoop (circumferential) and in the axial (longitudinal) directions of a closed end thin cylinder are usually expressed as below [7]:

$$\sigma_\theta \approx \frac{pr_i}{t}, \qquad \sigma_z \approx \frac{pr_i}{2t}$$

They are linear in p, meaning that under linear elastic conditions, the stress is proportional to the loading [7]. The equations above show that the relationship between pressure and stress is linear.

2.2. Lame solution (thick-walled cylinder relations)

For thick cylinders, stresses vary with the radial coordinate r∈[r_i,r_o]. When the pressure inside the cylinder P_i is greater than the outside pressure P_o, the internal pressure is greater than the outside pressure, causing the cylinder to yield (deform):

$$\sigma_r(r) = A - \frac{B}{r^2}, \qquad \sigma_\theta(r) = A + \frac{B}{r^2},$$

with constants A, B depending on the boundary conditions σ_r (r_i)=-p_i and σ_r (r_o)=-p_o. If p_o=0, then the common case is the case where p_o is set to 0:

$$\sigma_r(r) = \frac{p_i r_i^2}{r_o^2 - r_i^2}\left(1 - \frac{r_o^2}{r^2}\right), \qquad \sigma_\theta(r) = \frac{p_i r_i^2}{r_o^2 - r_i^2}\left(1 + \frac{r_o^2}{r^2}\right).$$

The hoop stress, σ_θ (r), is the largest at the inner wall, r=r_i, which is the most important part of the vessel when under pressure loading. Schneider and Kienzler make an explicit return to Lamé's problem and stress the plotting of stress distributions and maximum von Mises stress, making it clear that the overall maximum equivalent stress typically is at or near the inner surface.

2.3. von Mises stress (distortion energy based metric)

Since yielding and many integrity criteria depend on deviatoric (distortional) stress, the von Mises equivalent stress is used as a scalar measure derived from the principal stresses $\sigma_1, \sigma_2, \sigma_3$.

$$\sigma_{vM} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}.$$

In an axisymmetric pressurized cylinder, principal stresses are typically combinations of σ_r, σ_θ, and σ_z, so the large stress difference between high hoop stress and compressive radial stress near the inner wall increases σ_{vM} there, consistent with the simulated "inner-surface maximum" observed in this work [3].

3. Methodology

3.1. Simulation environment and solver

A linear static structural analysis was performed utilizing the cloud platform SimScale. The platform documentation indicates that its Static analysis type uses the Code_Aster solver to compute time-invariant displacements and stresses driven by applied loads and constraints.

3.2. Geometry

The geometry of the model is a three-dimensional hollow cylinder, which corresponds to a simplified part of a pressure vessel. The present study does not focus on a particular industrial configuration (such as heads, nozzles, or supports) but rather concerns the fundamental condition of internal pressure loading and the consequent stress distribution, which can be interpreted by classical theories of thin and thick walled cylinders. The geometry adopted allows a clear investigation of the behaviour of stress scaling and localization under internal pressure as shown in figure 1.

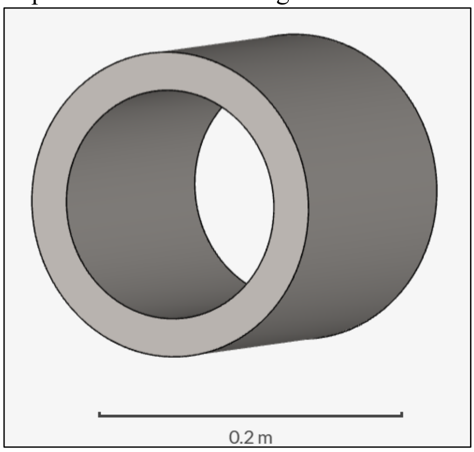

Fig. 1. 3-D shape of the cylindrical pressure vessel employed in the finite element analysis.

The model is a pressure vessel of cylindrical shape, having constant axial wall thickness. Correctly applies pressure loading and boundary conditions in the simulation, the inner and outer surfaces are explicitly defined.

3.3. Material model

The cylinder was assigned an isotropic, linear-elastic material model, which is also typical of the assumptions that are made in small-strain static analysis. To apply pressure-vessel stress screening or to compare against closed-form elastic formulas prior to considering plasticity and/or geometric nonlinearity, linear elastic modeling is a common baseline approach.

3.4. Boundary conditions

The load and restraint state were specified by two boundary conditions: The internal pressure was applied to the inside of the cylinder using the pressure boundary condition of the platform, which is a distributed load on the faces specified. The three simulated instances were p=1 MPa, 2 MPa and 3 MPa. To prevent rigid body motion, fixed support was used at one end of the cylinder. On the faces assigned to the platform, the platform's fixed support constraint makes all the translational degrees of freedom equal to zero. The same documentation warns of the possibility of localized stress concentrations and/or steep gradients in the region surrounding the constrained area – a Saint Venant type effect.

3.5. Meshing strategy

The domain was discretized using an automatically generated unstructured tetrahedral mesh consisting of approximately 1600 elements, representing a computationally efficient setup suitable for a preliminary linear-elastic analysis. For complicated geometries, unstructured tetrahedral meshes are typically used, but are sensitive to mesh density, and can result in differences in peak stress values compared to structured (mapped) meshes, even in cases when convergence-controlled conditions are applied. This underscores the importance of taking mesh sensitivity into account in finite element analysis, as shown in Figure 2.

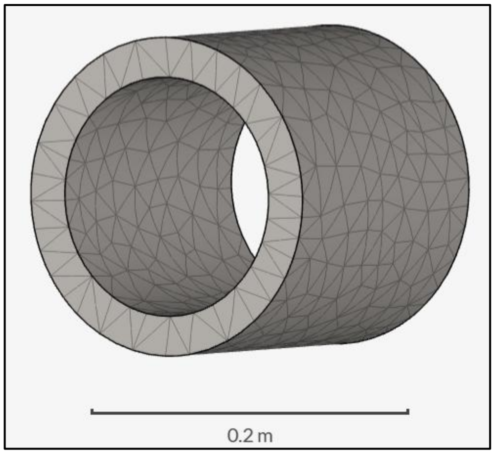

Fig. 2. Finite element mesh of the pressure vessel model with an unstructured tetrahedral discretization (~1600 elements).

The mesh properly represents the cylindrical geometry and the stress gradients especially close to the inner surface where stress concentration is expected.

3.6. Metric extracted and post processing

Post processing & extracted metrics. The von Mises stress field was plotted in the post-processing environment and the maximum value was reported for each load condition. Platform documentation states that von Mises stress is an equivalent stress computed from the principal stresses and associated with the distortion-energy theory of yielding, justifying its use as the primary scalar stress indicator.

4. Results

4.1 Stress results and distribution

Reported von Mises maxima from real SimScale runs Table 1 shows the maximum von Mises stress values obtained from the actual SimScale finite element simulations under varied internal pressure levels. The reported results are based entirely on numerical simulations, with no synthetic or assumed data introduced.

Table 1. FEA data of internal pressure vs peak von Mises stress

Internal pressure p (MPa)	Peak von Mises stress $\sigma_{vM,\max}$ (MPa)
1	4.35
2	8.70
3	13.06

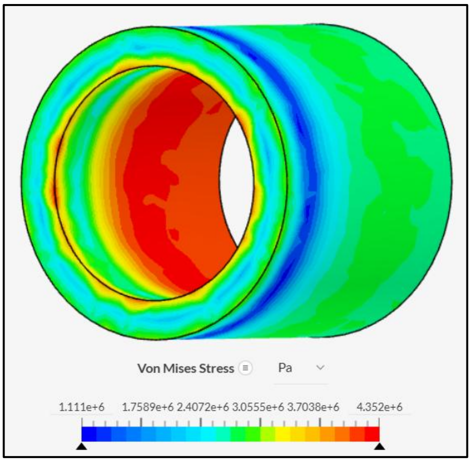

Fig. 3. The pressure vessel's von Mises stress distribution at internal pressure 1 MPa.

The stress field has a well defined maximum at the inner surface of the cylinder where the largest von Mises stress is found. This is just what the theory predicts. The higher the internal pressure the greater the circumferential (hoop) strains along the inner wall relative to the outer surface. The stress distribution corresponding to an internal pressure of 2 MPa is shown in Figure 4.

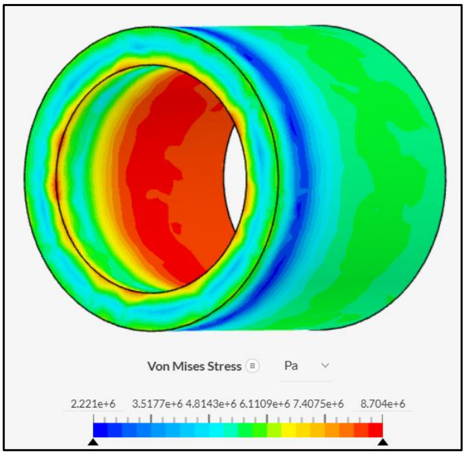

Fig. 4. von Mises stress distribution in the pressure vessel with internal pressure 2 MPa.

The results show a similar spatial distribution pattern as the case of 1 MPa with an increase in the magnitude of stress through the structure. The stress distribution at an internal pressure of 3 MPa is given in Fig. 5.

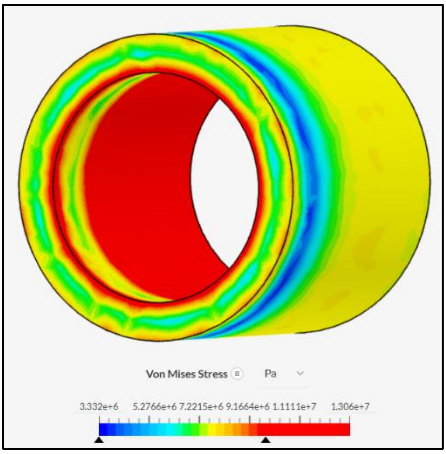

Fig. 5. von Mises stress distribution in the pressure vessel at an internal pressure of 3 MPa.

At maximum applied pressure the von Mises stress is maximum although the distribution pattern is the same as at lesser pressures. The stress concentration at the inner surface is more obvious and the total stress field grows accordingly, which further verifies the linear relation between the internal pressure and the stress.

4.2 Linear regression analysis

A linear regression of $\sigma_{vM,\max}$ against p yields:

$$\sigma_{vM,\max}(p) \approx 4.355\,p - 0.0067 \quad \text{(MPa)},$$

with coefficient of determination $R^2 \approx 0.9999996$. This near-perfect linear fit quantitatively confirms proportional stress scaling across the investigated pressure range.
The corresponding pressure–stress relationship is illustrated in Figure 6.

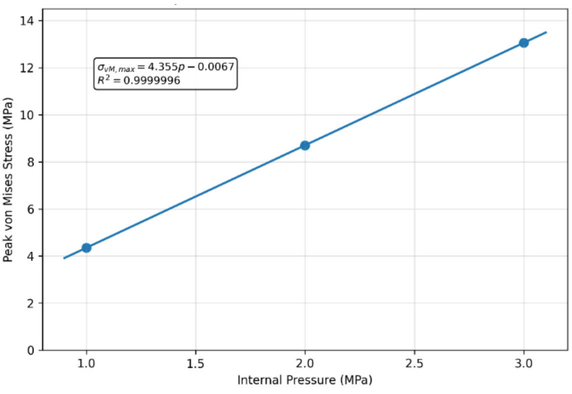

Fig. 6. Linear correlation of internal pressure with maximum von Mises stress from finite element simulations.

The pressure-stress data points are very close to the fitted regression line showing that there is an almost perfect linear relationship between the internal pressure and peak von Mises stress in the range that we have looked at. This is what we would expect from linear elasticity theory that states that the stress should scale proportionally to the applied load in the case of small deformations. The coefficient of determination (R^2~0.9999996) also shows that the finite element simulations are accurate and that there is very little numerical deviation.

Physically, this linear behavior is indicative of elastic deformation of the material over the range of pressures investigated (1 - 3 MPa) and there is no evidence of a nonlinear material response or of a geometric instability. The gradient of the regression line also provides an indication of the sensitivity of the vessel to the internal pressure and may be useful as an index in early design and safety assessment.

The results are in good agreement with the theoretical values, proving the reliability of the finite element model used, the boundary conditions, the mesh discretization and the material assumptions.

4.3 Stress-field characteristics and localization

In all pressure cases, the stress field has uniform spatial distribution of circumferential (hoop-driven) behavior typical of internally pressured cylinders. The maximum von Mises stress occurs at the inner surface, which agrees with thick-cylinder elasticity theory, in which, the hoop stress $\sigma_\theta (r)$ tends to increase as r approaches r_i, the maximum value occurs at $r=r_i$ [3].

Additionally, the neighborhoods around the fixed support have stress gradients. Although the boundary conditions can affect the distribution of stress at a local level, the results overall show that the boundary conditions do not have a significant influence on the final stress, and that the internal pressure effects are dominant..

5. Discussion

5.1 Linear pressure–stress relationship

This linear relationship $\sigma_{(vM,max)} \propto p$ is the signature of linear static structural response, which occurs when the material behavior is linear elastic, and the boundary conditions are proportionate to the pressure (scaled without change in its distribution): stresses scale with load magnitude. This is in accord with the "equilibrium" derived thin cylinder relations which are explicitly linear in p [7]. In real design processes, these proportionalities are used for parametric studies and a quick screen of scenarios prior to nonlinear phenomena (plasticity, large deformation, contact), that do not allow for strict linear scaling [8].

The regression intercept is essentially null compared to the magnitudes of the stresses (approx. -0.007 MPa) and the small deviation of the point at 3 MPa from the “triple” value may be due to rounding and to the fact that it was extracted from a maximum rather than from an unstructured mesh, which does not necessarily imply that there is physical nonlinearity. Small numerical offsets can also be added in linear analyses when using the mesh discretization and using “maximum” for nodal/element representations [9].

5.2 Inner-surface stress explanation

It is possible to explain the maximum stress at the inner surface both physically and by classical elasticity theory. Stresses from the inside press outward against the inside boundary, while stresses from the outside push inward against the outside boundary – called circumferential (hoop) stresses. The primary method of load bearing for cylindrical structures is through these stresses.

The Lamé solution of thick wall cylinder demonstrates that the hoop stress is maximum at the inner radius and in the theory is reduced to the outer surface. Also, the difference between the principal stresses affects the von Mises stress; the magnitude of the von Mises stress is amplified near the inner wall due to the high hoop stress and the compressive radial stress (equal to -p).

This combined impact causes the greatest von Mises stress to be always at the inner surface, in line with the analytical solutions and the finite element results given in this work.

5.3 Mesh influence and numerical credibility

An automated tetrahedral meshing, with about 1600 elements, was used for the analysis and this was found to be sufficient to capture the overall stress distribution and to show the proportional scaling behavior. But this mesh density is not enough for the peak stress to be mesh independent.

Known areas of local stress maximums such as in the vicinity of geometric boundaries and constrained regions are known to be sensitive to mesh resolution. Refined meshes can show up higher stresses in certain parts of the mesh, while coarse meshes can smooth out stress gradients. Also, for fixed boundary conditions, artificial areas of high stress can be created, which will impact peak stress evaluation.

Hence, a mesh convergence study is suggested to get mesh independent results. The results in the present work are to be regarded as indicative of the stress trends and localization, but not as a converged estimate of the peak stress values.

5.4 Engineering implications and comparison to broader literature

The results are a useful confirmation of the stress response of pressure vessels in the linear elastic range. The linear relationship between pressure and stress was observed and is consistent with classical design equations, and can be used to pre-evaluate the structural safety of the preliminary design by using the finite element analysis [7].

The maximum stress at the inner surface is in accordance with the thick-cylinder theory. This indicates the importance of bore areas for the evaluation of the structural integrity of a structure notably for failure due to fatigue and corrosion [6].

The present model is of course an idealization, and is a cylindrical shape. Nevertheless, the results are still relevant for real life where geometrical discontinuities such as nozzles and welds can lead to large increases in stress concentrations. These results justify therefore a step-wise design process beginning with simple models and then working towards more complex geometries [6]

6. Limitations and future work

6.1 Limitations

Limitations of the model are: (i) it assumes linear elastic material behavior, (ii) the end fixed support may create local gradients, (iii) it is a single coarse tetra mesh level (~1600 elements), and (iv) there is no explicit end closure, heads or nozzles in the model. The important point is that there is a known fixed-constraint effect in static analysis that must be used with care when interpreting maxima, and which often drives local refinement and/or alternative constraints.

6.2 Future work

There are several extensions that can further enhance the engineering usefulness of the present work. First, a mesh convergence study is recommended to ensure grid-independent stress predictions, particularly near the inner surface and constrained regions. Secondly, the nonlinear material behavior would allow elastoplastic analysis and the burst pressure prediction in a more realistic manner as shown in earlier works [11]. Third, the realistic features such as heads, nozzles and local wall thinning would make stress concentrations and critical fatigue areas easier to locate. Finally, the combination of internal pressure and temperature loading conditions would provide a more realistic depiction of the operation of vessels [10].

7. Conclusion

A cloud-based finite element study of a cylindrical pressure-vessel segment under internal pressure was performed using real SimScale outputs. For internal pressures of 1, 2, and 3 MPa, the computed peak von Mises stresses were 4.35, 8.70, and 13.06 MPa, respectively. These results establish a clear linear relationship between pressure and peak equivalent stress, quantified by a linear fit $\sigma_{vM,\max} \approx 4.355p - 0.0067$ (MPa) with $R^2 \approx 0.9999996$. Stress contours indicate that the maximum von Mises stress occurs at the inner wall, consistent with thick-cylinder elasticity in which hoop stress is maximal at the bore and the von Mises measure amplifies stress differences involving the compressive radial component at the inner surface. Some of the interpretational significance of the boundary-condition-induced gradients near fixed supports, and the requirement for mesh convergence to validate the accuracy of peak stresses in the study are also emphasized. Further studies should be conducted to extend the approach to mesh sensitivity, nonlinear elastoplastic response, and prediction of burst pressure as well as the incorporation of realistic vessel discontinuities (nozzles, heads, thinning) for which FEA has proven superior to analytical approximations.

References

1. K. Hazizi, M. Ghaleeh, Design and analysis of a typical vertical pressure vessel using ASME code and FEA technique. *Designs* **7**, 78 (2023). https://doi.org/10.3390/designs7030078
2. G.B. Sinclair, J.E. Helms, A review of simple formulae for elastic hoop stresses in cylindrical and spherical pressure vessels: What can be used when. *Int. J. Press. Vessel. Pip.* **128**, 1-7 (2015). https://doi.org/10.1016/j.ijpvp.2015.01.006
3. P. Schneider, R. Kienzler, Dimensioning of thick-walled spherical and cylindrical pressure vessels. *Math. Mech. Solids* **25**, 1391-1404 (2017). https://doi.org/10.1177/1081286517713243
4. M. Bozkurt, D. Nash, A. Uzzaman, A comparison of stress analysis and limit analysis approaches for single and multiple nozzle combinations in cylindrical pressure vessels. *Int. J. Press. Vessel. Pip.* **194**, 104563 (2021). https://doi.org/10.1016/j.ijpvp.2021.104563
5. X. Chen, S. Fang, H. Chen, Stress concentration factor and fatigue analysis of a lateral nozzle with local wall thinning. *Engineering Failure Analysis.* **105**, 289-304 (2019). https://doi.org/10.1016/j.engfailanal.2019.07.004
6. K. Tserpes, I. Sioutis, G. Floros, E. Moutsompegka, Numerical simulation of debonding of a composite-to-metal adhesive joint subjected to centrifugal load. *Eng. Fail. Anal.* **136**, 106131 (2022). https://doi.org/10.1016/j.engfailanal.2022.106131

7. A. Ibrahim, Y. Ryu, M. Saidpour, Stress analysis of thin-walled pressure vessels. *Modern Mechanical Engineering.* **5**, 1-9 (2015). https://doi.org/10.4236/mme.2015.51001
8. A. Mohan, S. Julyes Jaisingh, C.P. Goldin Priscilla, Effect of autofrettage on the ultimate behavior of thick cylindrical pressure vessels. *Int. J. Press. Vessel. Pip.* **194**, 104546 (2021). https://doi.org/10.1016/j.ijpvp.2021.104546
9. A. Haider, A comparison of free and mapped meshes for static structural analysis. *Fatigue Aircr. Struct.* **15**, 115-132 (2024). https://doi.org/10.2478/fas-2023-0007
10. A.O. Abdelatif, J. Ožbolt, A. Omara, Stress distribution in thick-walled cylinder due to non-uniform radial pressure on the example of reinforcement corrosion in concrete. *Mater. Corros.* **71**, 1660-1666 (2020). https://doi.org/10.1002/maco.202011661
11. J. Chai, J. Zhong, B. Xu, Z. Zhang, Z. Shen, X. Zhang, J. Shen, Finite element analysis and experimental validation of the failure characteristic of pressurized cylinder. *International Journal of Structural Integrity.* **14**, 874-890 (2023). https://doi.org/10.1108/IJSI-06-2023-0055
12. H. Fan, L. Hu, Pressure vessel nozzle local stress prediction software based on ABAQUS–machine learning. *SoftwareX* **24**, 101550 (2023). https://doi.org/10.1016/j.softx.2023.101550
13. Bhilare, S. L., Hinge, G. A., & Kumbhalkar, M. A., Experimental investigation for the measurement of pressure and discharge through modified gate valve. *Larhyss Journal, 57*, 121–144, (2024)..
14. W.R. Johnson, X.-K. Zhu, R. Sindelar, B. Wiersma, A parametric finite element study for determining burst strength of thin and thick-walled pressure vessels. *Int. J. Press. Vessel. Pip.* **204**, 104968 (2023). https://doi.org/10.1016/j.ijpvp.2023.104968
15. M.A. Khorshidi, D. Soltani, Analysis of functionally graded thick-walled cylinders with high order shear deformation theories under non-uniform pressure. *SN Appl. Sci.* **2**, 1362 (2020). https://doi.org/10.1007/s42452-020-3179-0
16. E.S. Kim, Structural integrity evaluation of CNG pressure vessel with defects caused by heat treatment using numerical analysis. *J. Mech. Sci. Technol.* **33**, 5297-5302 (2019). https://doi.org/10.1007/s12206-019-1021-7