



The static and modal analysis of concrete tank filled with water

Alexander Filip, Katarína Tvrďá, Mária Minárová
Slovak University of Technology, Faculty of Civil Engineering, Bratislava, Slovakia
info@alexanderfilip.com
katarina.tvrdav@stuba.sk, <https://orcid.org/0000-0003-4437-3127>
maria.minarova@stuba.sk, <https://orcid.org/0000-0001-7347-1047>



Citation: Filip, A., Tvrďá, K., Minárová, M., Tank analysis under load effects, *Fracture and Structural Integrity*, 74 (2025) 217-226.

Received: 11.07.2025

Accepted: 15.08.2025

Published: 04.09.2025

Issue: 10.2025

Copyright: © 2025 This is an open access article under the terms of the CC-BY 4.0, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

KEYWORDS. Tank, Computational model, Static analysis, Modal analysis, Natural frequency, Mode shape.

INTRODUCTION

In modern civil engineering, tanks and reservoirs play essential role in ensuring the reliable storage of liquids, including drinking water, industrial liquids, and even gases. With the increasing requirements for safety and efficiency in the use of water and energy resources, the importance of static and dynamic investigations of these special structures is still increasing. Numerical modelling using the Finite Elements Method (FEM) is one of the most effective tools for analysing complex tank-liquid interactions. Similar finite element investigations have been reported for the above-ground steel storage tanks under the static and the seismic loads by Salem et al. [20]. Additionally, the computational methods using FEM can be efficiently employed for the failure pressure prediction in cylindrical pressure vessels with high accuracy, often achieving prediction errors below 1% [7]. Modern ANSYS APDL techniques offer sophisticated approaches for linear



elastic fracture mechanics analysis, which can be particularly useful for assessing structural integrity of tanks with potential crack-like defects [23]. The Ansys Mechanical environment provides comprehensive fracture simulation capabilities, allowing engineers to use fracture mechanics principles to simulate crack effects in structural components [5]. These computational tools have been successfully applied to various pressure vessel applications, including reactor pressure vessels and storage tanks [3].

In this work, the focus is put both on a detailed analysis of the behavior of the cylindrical reinforced concrete tank filled with water using the Ansys software package – Workbench (WB) and Mechanical APDL (M/APDL). The comparison of numerical and analytical calculations brings useful information about the accuracy and power of various approaches to modelling structural and fluid behavior.

There are many studies in literature devoted to the static and dynamic analysis of the structures. For dynamically loaded members, the modal analysis has to be carried out first, in order to assess the natural frequencies and mode shapes that have essential impact to the mechanical behavior under the dynamic load. E.g., Koubova [14] offers such stipulation of natural frequencies and mode shape stipulation for a curved beam. Similarly, for the cylindrical tanks, the modal analysis with various factors, such as configuration, ambience, etc., influencing their mechanical behavior are widely dealt with. The stability analysis under the seismic load investigation is described in Jerath and Lee [11], who studied the buckling of the tank during earthquake typical for California, Kotrasová and Kormaníková [13] focus to the seismic response of cylindrical fluid filling tanks fixed to rigid foundations, etc. The reports of Finite element (FE) approaches in Ansys Workbench can be found e.g. in Rodwal [18], where the different types of layered soil under the tank are considered and its impact on the tank’s behavior resting on the ground is discussed. In Tănase [22], we can find the comparison of the stress state determination of the thin-walled structures with both the membrane theory analysis and FE method. In [4], Amabili et al. bring an experimental validation of the modal parameters calculated theoretically before and explores the impact of hydrostatic pressure to the natural frequencies. Virella et al. in [24] offer a case study in which he examines the tank subjected to a horizontal vibration, the results for various aspect ratios, i.e. ratio height/diameter, are dealt, resulting in changes of the eigenvalues, natural frequencies, and mode shapes. The discrete impulsive-convective model for tanks with manholes can be found in Zanni et al. [25]. The various tank geometry and critical fill levels are tested as well as the fluid-soil-structure interaction. The seismic safety of industrial liquid storage and sloshing is inspected in Kangda [12]. Ruan et al. [19] reveals the significance of the computational fluid dynamics involvement in order to enhance capabilities for analyzing complex tank behavior, particularly in pressurized thermal shock scenarios where both thermal and mechanical effects have to be considered.

The long-term degradation mechanisms, such as reinforcement corrosion, can affect the stress distribution, as demonstrated by Miloudi et al. [17] and Bouzelha et al. [6] for elevated concrete tanks. The adoption of fatigue and fracture mechanics-based approaches characteristic for pressure vessels (Majid et al. [15]), can play a significant role, too; such approaches increase the plausibility of the employed numerical methods.

ANALYSIS OF THE CYLINDRICAL TANK – THEORETICAL BACKGROUND

Hydrostatic pressure

In the mechanical problems of building physics, it is necessary to take all essential loads into account. Within the study of cylindrical tanks and reservoirs, along with the self-weight of the structure and the liquid, the effect of the structure-liquid interaction, both static and dynamic, has to be taken into account. Accordingly, the effect of hydrostatic pressure on the walls of the tank is included in the investigation and computation [10].

Generally, the hydrostatic pressure is given by the formula (1) according to Sobota [21].

$$p(\xi) = \rho g (b - \xi) \tag{1}$$

where p is the hydrostatic pressure of the liquid [Pa], ρ is the density of the liquid [kg/m³], g is the gravitational acceleration [m/s²], ξ is the height of the surface of the liquid inside the tank (above the ground) [m], b is the height of the tank [m].

It is a known fact, evident from the formula (1) as well, that the hydrostatic pressure increases with depth of the liquid, hence the term *hydrostatic triangle*, see Fig. 1 left, is often used herein. The hydrostatic pressure magnitude affecting the inner surface of the tank is depicted in Fig. 2.

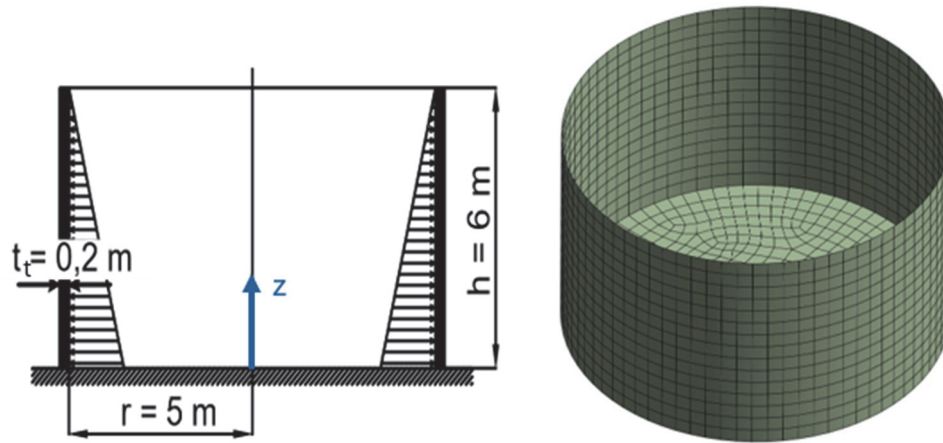


Figure 1: A schematic view of the analyzed tank and a discretized shell geometry.

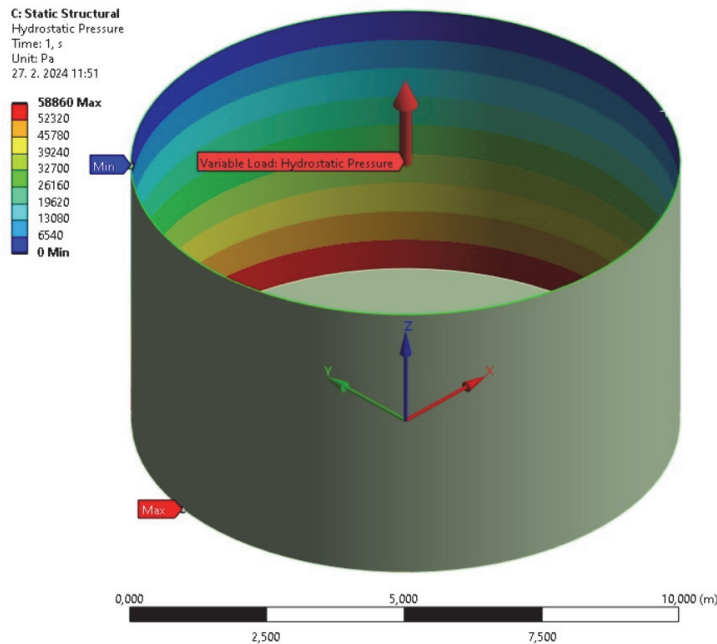


Figure 2: Hydrostatic pressure distribution in a fully filled tank, ANSYS graphical output.

Dynamic analysis

As far as the interaction between the tank structure and the liquid is concerned, the dynamic influences prevalently dominate over the static ones.

During dynamic analysis, we trace the effect of the load varying in time, and its mechanical impact, the resulting deformation, stress and inner forces depending not only on this load, but on the inertial forces, too. Let us recall the inertial forces act in the direction opposite to the acceleration vector. Naturally, the deformations vary with time as well. The differential equation governing the dynamic effects on a structure is given by the formula (2).

$$\mathbf{M}\ddot{\mathbf{v}}(t) + \mathbf{C}\dot{\mathbf{v}}(t) + \mathbf{K}\mathbf{v}(t) = \mathbf{F}(t) \quad (2)$$

where \mathbf{M} is the mass matrix, \mathbf{C} is the damping matrix, \mathbf{K} is the stiffness matrix, $\mathbf{F}(t)$ is the load vector, $\mathbf{v}(t)$ is the vector of displacements, $\dot{\mathbf{v}}(t)$ is the velocity vector, $\ddot{\mathbf{v}}(t)$ is the acceleration vector.



Modal analysis

Modal analysis provides a general insight into the undamped mechanical response of a structure, which is not exciting by a force. It studies the natural frequencies and mode shapes of the structure, during free vibration. Herein, we get the governing Eqn. (2) directly from the Eqn. (3), putting zero damping factor \mathbf{C} and zero exciting force $\mathbf{F}(t)$.

$$\mathbf{M}\mathbf{v}(t) + \mathbf{K}\mathbf{v}(t) = 0 \tag{3}$$

The non-trivial solution of the Eqn. (3) is then determined by the nonhomogeneous attached initial conditions – either the non-zero initial deflection or non-zero initial velocity.

Supposing the harmonic vibration (4)

$$\mathbf{v}(t) = \bar{\mathbf{v}} e^{i\omega t} \tag{4}$$

and supplying to (3) we get (5):

$$(\mathbf{K} - \omega^2 \mathbf{M})\bar{\mathbf{v}} = 0 \tag{5}$$

where $\bar{\mathbf{v}}$ is the amplitude vector of harmonic periods and ω is natural frequency. The Eqn. (5) has its non-trivial solution for discrete values of natural frequencies $\omega_i, i = 1, n$, with n being the order of matrices \mathbf{K} and \mathbf{M} . Natural frequencies are computed under the condition of zero determinant (6).

$$\det|\mathbf{K} - \omega^2 \mathbf{M}| = 0 \tag{6}$$

Each natural frequency ω_i corresponds to a particular mode shape of vibration $\bar{\mathbf{v}}_i$. In FEM systems, a range of algorithms is typically used to solve this problem — for instance; ANSYS offers seven algorithms applicable to different cases.

The objective of modal analysis is to obtain the fundamental dynamic characteristics of the analyzed structure serves to avoiding of the resonance during life span of the construction. Moreover, modal analysis is the starting point for solving numerous other tasks.

Many mode shapes can be achieved by calculations on more complex structures, but only a few of them are significant. Generally, the lower the natural frequency, the more significant the mode shape. However, we can also evaluate them quantitatively using the so-called participation factor. It expresses the degree of participation of its mode shape on the displacements and stresses of the structure in a certain direction (7),

$$\gamma_i = -\bar{v}_i^2 \mathbf{M} \mathbf{d} \tag{7}$$

where γ_i is the i^{th} coefficient of the contribution in the direction of the vector excitation \mathbf{d} , \mathbf{M} is a mass matrix of the structure. Another useful indicator is the effective mass $\mathbf{M}_{eff,i}$, which is understood as the mass of the structure excited by the movement of the i^{th} mode shape in a certain direction [7]. This one is given in kilograms and is obtained by a simple calculation, as it is in the Eqn. (8).

$$\mathbf{M}_{eff,i} = \gamma_i^2 \tag{8}$$

STATIC AND MODAL ANALYSES OF THE CYLINDRICAL TANK

In this chapter, the results of the static and modal analyses are presented. The comparison of the results obtained from Ansys Workbench and Ansys APDL with existing analytical solutions is provided and discussed.

The first comparison intended, is an evaluation of the agreement between the Ansys computations and analytical results in a simple hydrostatic calculation and in a modal analysis. For the static analysis, within the Workbench platform, a function

of hydrostatic pressure was employed. The results obtained were then compared with the analytical calculations from Sobota [21] and, moreover, with the numerical calculations Haladej [8], where the author uses the method of the hydrostatic triangle (TRIAN) within the Ansys platform M-APDL, using the FLUID30 element type.

The Ansys Workbench package was used to model our tank fully filled with water, both in the case of WB;LIN and WB;QUAD. In the WB;LIN calculations, the linear acoustic elements FLUID30 were used for water and air. In the WB;QUAD calculations, the quadratic acoustic elements FLUID220 (cube/pyramid/prism) and FLUID221 (tetrahedron) were used for water and air. As in the aforementioned work by Haladej [8] so in ours, we used shell elements to model the tank itself.

Geometry

All computations presented in this paper were carried out on a thin-walled (shell-type) cylindrical tank with a base radius $r = 5$ m, wall height $b = 6$ m, and wall thickness $t = 0.2$ m. Fixed support at the foundation was assumed as boundary condition.

Material characteristics

The focused cylindrical tank is made of reinforced concrete, with a Young's modulus of elasticity $E = 21$ GPa and Poisson's ratio $\nu = 0.167$. It is fully filled with water, with a density $\rho = 1000$ kg/m³, the value of gravitational acceleration 9.81 m/s² is taken.

Static analysis of the focused tank

As mentioned above, in addition to the self-weight of the tank filled with water, hydrostatic pressure also acts on the structure. Due to this hydrostatic pressure applied to the cylindrical tank, the so-called "elephant foot" shape appears. Moreover, this special deformation effect is monitored in the static analysis, focused on the deflection of the walls of an axially symmetric tank, see Fig. 3.

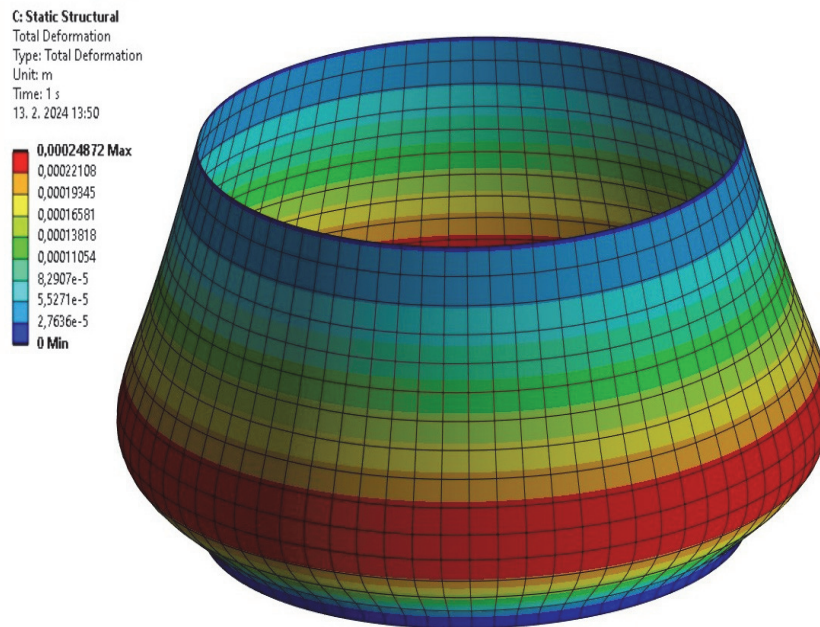


Figure 3: Static analysis: deformations caused by hydrostatic pressure.

Meter by meter along the height, the computed horizontal deformations in the direction of the outer normal are then taken. The results of the individual analysis are summarized in Tab. 1. All deformations are given in millimetres. Our computation was carried out in ANSYS Workbench software; the results are provided in the last two columns of Tab. 1, denoted by WB. The hydrostatic pressure was employed in both method WB. In the case WB;LIN, the tank was modelled by the linear shell elements SHELL181 (4-nodes), in the case WB;QUAD the second other shell SHELL281 (8 nodes) was used. Our numerical results from ANSYS Workbench are then compared with analytical ones and the with two numerical ANSYS Mechanical APDL outputs of Haladej [8], see the third and fourth columns in Tab. 1. In M/APDL TRIAN, the hydrostatic



pressure was modelled by using the hydrostatic triangle method, in M/APDL;FLUID30 by using the acoustic elements FLUID30.

Height ζ [mm]	Analytically [21]	M/APDL; TRIAN [8]	M/APDL; FLUID30 [8]	WB; LIN	WB; QUAD
0	0.000	0.000	0.000	0.001	0.000
1000	0.191	0.209	0.210	0.186	0.195
2000	0.249	0.261	0.262	0.251	0.248
3000	0.188	0.189	0.189	0.188	0.187
4000	0.120	0.122	0.121	0.119	0.119
5000	0.059	0.061	0.060	0.058	0.059
6000	0.000	0.000	0.000	0.000	-0.008

Table 1: Static analysis – Deformations in the outer normal direction due to the hydrostatic pressure.

In the ANSYS Workbench platform, the walls of the tank were discretized using either LIN – linear shell elements (SHELL181- 4 nodes), or QUAD – quadratic shell elements (SHELL281 -8 nodes). The geometric model consists of 1741 shell quadrilateral elements, with approximate edge longitude of 0.4 m. We have verified that by a subsequent refinement of the FE net, the results are précised only negligibly, and the number of elements is optimal. On the other hand, Haladej [8] uses 3792 elements of type SHELL43.

Statistical validation

The accuracy validation is based on the following statistical calculation by a recently developed methodology. The data are assessed successively. First, one by one, the deflections in the direction of outer normal of particular level are compared with reference values. Taken the differences from the reference values per unit height, the so-called relative error E_r arises, given by the formula (9).

$$E_r(f_i, g_i) = 100 \frac{f_i - g_i}{f_i} \tag{9}$$

in which f_i is the i^{th} level of height – the value yielded based on a numerical method and g_i is the i^{th} reference value – the value yielded analytically.

We assess the accuracy of individual model based on the mean absolute percentage error ($MAPE$) according to (10), from Statistics How to [16]:

$$MAPE(f, g) = \frac{100}{N} \sum_{i=1}^N \left| \frac{f_i - g_i}{f_i} \right| \tag{10}$$

In which f stands as an identifier of the used numerical method, g is the reference value, N is the number of measurements, $N = 7$ in our case. Mean Absolute Percentage Error $MAPE$ is a statistical measure that helps us to determine how accurate our results from numerical calculations are in relation to analytical values. By comparing the $MAPE$ of different models, we can evaluate which model performs better in terms of accuracy. Lower values of $MAPE$ indicate higher accuracy, while higher values indicate lower accuracy.

The relative errors (9) and $MAPE$ values (10) for the particular tools are presented in Tab. 2.

From the $MAPE$ values presented in Tab. 2, we can briefly conclude that WB:QUAD works best. It can be observed that the results obtained from Workbench are generally better than those from other methods presented here.

As expected, using quadratic elements provides results that are more accurate. On the other hand, it is somehow surprising that the calculations in ANSYS Workbench, when considering the hydrostatic triangle load (WB), achieved more accurate results than the calculation with the acoustic elements (FLUID30) in the ANSYS Mechanical APDL environment (M/APDL;FLUID30) as it is typical in static analysis. This question can stand as an object of further exploration involving a case study.

Height of the tank [mm]	M/APDL; TRIAN	M/APDL; FLUID30	WB; LIN	WB; QUAD
0	0.00%	0.00%	N/A	0.00%
1000	8.61%	9.05%	-2.55%	2.10%
2000	4.60%	4.96%	0.96%	-0.27%
3000	0.53%	0.53%	-0,02%	-0.57%
4000	1.64%	0.83%	-1.16%	-0.67%
5000	3.28%	1.67%	-0.99%	-0.41%
6000	0.00%	0.00%	N/A	N/A
<i>MAPE</i>	2.67%	2.43%	1.14%	0.67%

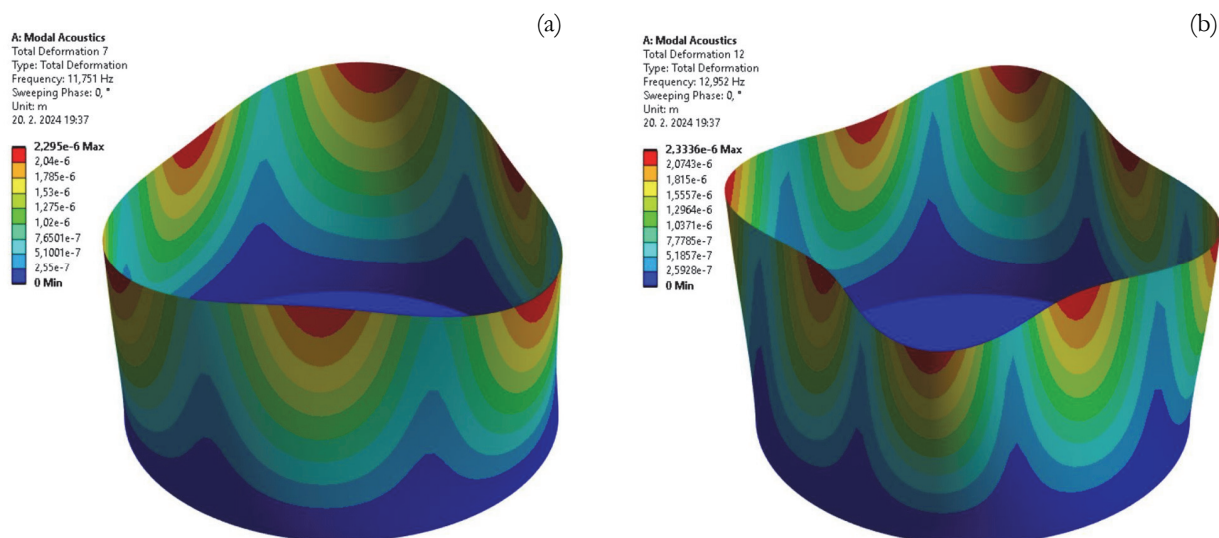
Table 2: Static analysis – Relative errors of the numerical methods (hydrostatic pressure involved).

Modal analysis of the focused tank

As mentioned above, since the natural frequencies strongly affect the mechanical response of a construction to a dynamic load, the modal analysis should precede each dynamic analysis. The modal analysis theory is based on the fact that each body has a spectrum of natural frequencies, which depend on the number of degrees of freedom with which it can vibrate. From a physical point of view, these frequencies correspond to the exchange of energy between different forms; in this case, vibrational energy is converted to kinetic energy, Harish [9]. Advanced numerical techniques using finite element methods have been successfully applied to study the seismic behaviour of unanchored steel tanks, particularly focusing on uplift phenomena that can significantly affect structural integrity [2]. These computational approaches provide valuable insights into dynamic tank r under various loading conditions.

The frequency at which natural resonance occurs is called the eigenfrequency, and the corresponding shape of the vibrating body is called the mode shape. The first four mode shapes for fully filled tank, graphical output from Ansys Workbench can be seen in Fig. 4.

The comparison of two different tools (M/APDL and WB;QUAD) performance is presented in Tab. 3, the first four eigenfrequency solutions related to the first four mode shapes are issued. The data are provided for both empty and fully filled tank. In M/APDL approach [8], the liquid is modelled by FLUID30 element and tank walls by SHELL43 with linear approximating polynomial, while in our investigation using WB, quadratic element SHELL281 and hydrostatic pressure are employed. Since the results are similar across the various methods used, we can briefly conclude that the model is valid and that the results are physically consistent.



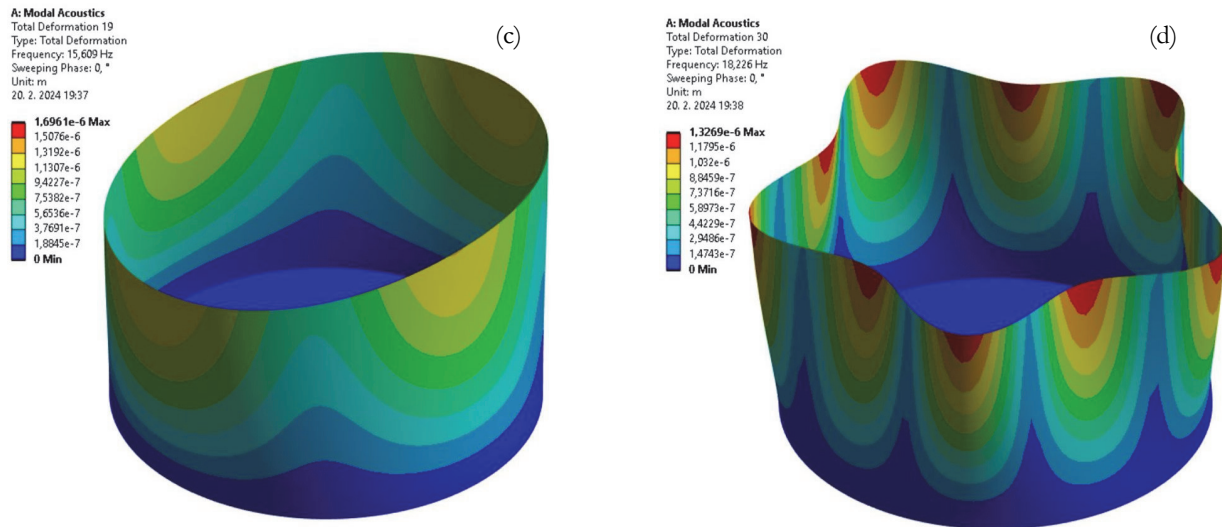


Figure 4: Modal analysis: the first four natural shapes a), b), c), d) of the fully filled tank. Output from WB;QUAD.

Mode shape	Empty tank M/APDL [8]	Empty tank WB; QUAD	Fully filled tank M/APDL [8]	Fully filled tank WB; QUAD
a	20.38 Hz	19.38 Hz	12.00 Hz	11.75 Hz
b	21.49 Hz	20.96 Hz	13.36 Hz	12.95 Hz
c	28.96 Hz	28.40 Hz	15.83 Hz	15.61 Hz
d	29.29 Hz	29.40 Hz	19.02 Hz	18.23 Hz

Table 3: Modal analysis – The first four eigenfrequencies of the empty a fully filled tank, computed by M/APDL aWB;QUAD.

CONCLUSION

The study presented in this paper provides a comprehensive overview of the static and modal analysis of a rotationally symmetrical tank made of reinforced concrete filled with water. The use of different approaches in the ANSYS Workbench and M-APDL environments offers detailed analysis of the influence of hydrostatic pressure on the wall deformation and the vibration frequency characteristics.

The results show a good level of agreement between the analytical calculations and the numerical simulations. The suitability of the given model and the accuracy of the calculations achieved were assessed, using the mean absolute percentage error. Lower MAPE values indicate higher accuracy, while higher values indicate lower accuracy. As expected, the use of quadratic elements reduces computational error. It is worth emphasizing the results of the modal analysis, which reveal the importance of the eigenfrequencies knowledge in order to future ensuring the safety and the reliability of these structures to resonance. The data obtained are relevant to engineering practice, especially when designing safe water-storing structures exposed to the loads and external influences that obviously vary in time.

Moreover, the recent developments in fracture mechanics analysis using advanced computational techniques, including virtual crack closure techniques and p-refinement finite element methods, offer promising avenues for enhanced structural integrity assessment of cylindrical shells and pressure vessels [1]. These methodologies provide efficient approaches for the damage-tolerant design strategies in various engineering applications, from aerospace to marine structures. This investigation points out to the fact that properly selected computational approach can efficiently model the real behavior of the structure. Hence, it is an essential tool for designing and optimizing similar engineering systems.



ACKNOWLEDGEMENTS

Supported by the Grants KEGA 030STU-4/2023, VEGA 1/0155/23 and VEGA 1/0036/23.

REFERENCES

- [1] Ahn, J.S. (2025). Computationally Efficient p-Version Finite Element Analysis of Composite-Reinforced Thin-Walled Cylindrical Shells with Circumferential Cracks, *Materials*, 18(7). DOI: <https://doi.org/10.3390/ma18071404>.
- [2] Akbari, J., Salami, O., Isari, M. (2020). Numerical investigation of the seismic behavior of unanchored steel tanks with an emphasis on the uplift phenomenon, *Frattura ed Integrità Strutturale*, 14(53), pp. 92–105. DOI: <https://doi.org/10.3221/IGF-ESIS.53.08>.
- [3] Alvarez-Loya, I., López-Grijalba, Y., Carbajal-Figueroa, L.G., Hernández-Gómez, L.H., Ruiz-López, P., Beltrán-Fernández, J.A. (2019). Lineal Elastic Fracture Mechanics Analysis Applied for the Evaluation of the Structural Integrity of a Boiling Water Reactor Vessel Considering Neutronic Irradiation, *Defect and Diffusion Forum*, 390, pp. 151–160.
- [4] Amabili, M., Pagnanelli, F., Pellegrini, M. (2001). Experimental modal analysis of a water-filled circular cylindrical tank, *Transactions on the Built Environment*, 56, pp. 267–276.
- [5] ANSYS Inc. (2021). Fracture Simulation with Ansys Mechanical, Ansys Webinar.
- [6] Bouzelha, K., Amazouz, L., Miloudi, N., Hammoum, H. (2019). Temporal analysis of the performance of a RC storage tank considering the corrosion., *Procedia Structural Integrity*, 22, pp. 259–266.
- [7] Chai, J.-H., Z.J.-P., X.B., Z.Z.-J., S.Z., Z.X.-L. and. (2023). Finite element analysis and experimental validation of the failure characteristic of pressurized cylinder, *International Journal of Structural Integrity*, 14(6), pp. 874–890.
- [8] Haladej, M. (2011). Tower reservoir modeling. Bratislava, Slovakia.
- [9] Harish, A. (2024). What is modal analysis and why is it necessary? SimScale. Blog. Available at: <https://www.simscale.com/blog/what-is-modal-analysis/>.
- [10] Hydrostatic Pressure. (n.d.). Available at: <https://www.geeksforgoeks.org/physics/hydrostatic-pressure/>.
- [11] Jerath, S., Lee, M. (2015). Stability Analysis of Cylindrical Tanks under Static and Dynamic Load Effects, *Journal of Civil Engineering and Architecture*, 9(1), pp. 72–79. DOI: <https://doi.org/10.17265/1934-7359/2015.01.009>.
- [12] Kangda, M.Z. (2021). An approach to finite element modeling of liquid storage tanks in ANSYS: A review, *Innovative Infrastructure Solutions*, 6(4). DOI: <https://doi.org/10.1007/s41062-021-00589-8>.
- [13] Kotrasova, K., Kormanikova, E. (2017). The Study of Seismic Response on Accelerated Contained Fluid, *Advances in Mathematical Physics*, 2017, pp. 1–9. DOI: <https://doi.org/10.1155/2017/1492035>.
- [14] Koubova, L. (2024) Numerical Solution of Natural Frequencies and Mode Shapes, *Procedia Structural Integrity*, 63, pp. 34-42. DOI: <https://doi.org/10.1016/j.prostr.2024.09.006>.
- [15] Majid, F., Nattaj, J., Elghorba, M. (2016). Pressure vessels design methods using the codes, fracture mechanics and multiaxial fatigue, *Frattura ed Integrità Strutturale*, 10(38), pp. 273–280. DOI: <https://doi.org/10.3221/IGF-ESIS.38.37>.
- [16] Mean Absolute Percentage Error (MAPE)- Statistics How To. (n.d.). Available at: <https://www.statisticshowto.com/mean-absolute-percentage-error-mape/>.
- [17] Miloudi, N., Bouzelha, K., Hammoum, H., Aoues, Y., Amiri, O. (2021). Temporal analysis of the performance of an elevated concrete tank considering the corrosion of the steel reinforcement, *Frattura Ed Integrità Strutturale*, 15(56), pp. 94–114. DOI: <https://doi.org/10.3221/IGF-ESIS.56.08>.
- [18] Rodwal, V. (2019). Static and Dynamic Analysis of Water Tank by FEM: A Review, *J Emerg Technol Innov Res*, 6(3), pp. 500–507.
- [19] Ruan, X., Nakasuji, T., Morishita, K. (2018). An Investigation of the Structural Integrity of a Reactor Pressure Vessel Using Three-Dimensional Computational Fluid Dynamics and Finite Element Method Based Probabilistic Pressurized Thermal Shock Analysis for Optimizing Maintenance Strategy, *J Press Vessel Technol*, 140(5). DOI: <https://doi.org/10.1115/1.4040698>.
- [20] Salem, T., Maaly, H., Abdelbaset, A. (2021). Analysis of above-ground steel storage tanks resting over piles or stone columns, *Frattura ed Integrità Strutturale*, 15(57), pp. 40–49. DOI: <https://doi.org/10.3221/IGF-ESIS.57.04>.
- [21] Sobota, J. (1980). Structural mechanics. Part 2, Bratislava, Alfa, publishing house of technical and economic literature, Construction literature edition.



-
- [22] Tănase, M. (2021). Stresses Calculation for a Vertical Storage Tank: Membrane Theory vs Finite Element, Romanian Journal of Petroleum & Gas Technology, 2/2021(2), p. 8.
- [23] Tensor Engineering. (2024). Linear Elastic Fracture Mechanics with ANSYS APDL, Technical Documentation.
- [24] Virella J. C., Godoy L.A., Suárez L.E. (2006). Fundamental modes of tank-liquid systems under horizontal motions, Eng Struct, 28(10), pp. 1450–1461.
- [25] Zanni, A.A., Spyridis, M.S., Karabalis, D.L. (2020). Discrete model for circular and square rigid tanks with concentric openings – Seismic analysis of a historic water tower, Eng Struct, 211.
DOI: <https://doi.org/10.1016/j.engstruct.2020.110433>.