Numerical study of the stress-strain state of the vertical four-compartment cylindrical tank

S Saiyan and A Paushkin
Moscow State University of Civil Engineering, National Research University, 26 Yaroslavskoe shosse, Moscow, 129337, Russia
E-mail address: berformert@gmail.com

Abstract. A numerical study of the stress-strain state of the construction of the vertical four-compartment cylindrical tank at the stage of operation for storing various substances has been carried out. The purpose of the finite element calculation is to determine the optimal value of the thickness of its load-bearing walls and partitions, which ensure the mechanical safety of the construction under different modes of filling the containers. The reliability of the calculation results, which are carried out both in linear and geometrically nonlinear formulations, is provided by calculations in two software packages SCAD and ANSYS Mechanical APDL with comparison of solutions.

1. Introduction
Thin-walled cylindrical shells are widely used in a wide variety of areas of modern technology: in industrial and civil construction, mechanical engineering, shipbuilding, aircraft and rocketry, etc. Such systems are used in the form of silo buildings for the storage and processing of bulk materials, in port hydraulic construction, in construction such as reactors, etc. [1]. Multi-chamber tank systems provide the flexibility to store bulk solids substance, gases or liquids by allowing multiple different substances to be stored simultaneously in a single tank.

There are various types of intersection of construction elements and other constructional features, leading to the fact that calculations of this kind are presented due to the presence of local disturbances of the stress-strain state [2-5]. The resulting local stresses and deformations can reach significant values, so a thorough study of the behavior of multi-compartment constructions obtained by joining shells into a single construction is a very difficult task.

Modern analytical methods of calculation do not allow calculations the stress-strain state of multi-compartment constructions due to the presence of geometric features (changes in the cross-section of structural elements, hatches, load-bearing partitions, holes, etc.) which affect stresses and deformations in local zones [6]. The finite element method makes it possible to determine the stress-strain state of the elements of cylindrical tanks of various configurations, allowing us to take into account all local effects [7-9].
2. Research

2.1. Research task
Within the framework of this article, the calculations of the stress-strain state of a four-compartment vertical cylindrical tank under the action of hydrostatic pressure under various operating modes are carried out. The calculations were carried out in a linear and geometrically nonlinear formulation.

The calculated four-chamber tank is 9.5 m high, 5.5 m in diameter, in which the liquid is poured up to the 8.5 m mark. The tank has the following operating modes: filling one, two, three and all sections of the tank (figure 1).

2.2. Finite element models
The cylindrical vertical tank with inner walls for storing liquids is the system of outer walls and inner partitions that create sealed compartments for storing substances. The developed finite element models of a cylindrical reservoir are shown in figure 2. The calculations used the finite element software packages ANSYS Mechanical APLD and SCAD 21.1.

![Figure 1. Geometric model of a cylindrical vertical tank with internal walls for liquid storage.](image)

The purpose of the finite element calculation is to determine the optimal value of the thickness of the bearing walls and partitions of the cylindrical tank, which, together with the rest of the characteristics of the construction, provide mechanical safety and tightness of the construction.
Figure 2. Finite element models of a cylindrical vertical tank with internal partitions for storing liquids in the software packages ANSYS Mechanical APDL (left) and SCAD (right).

The calculations used finite elements of the shell type (Type 44 in the SCAD and SHELL181 in the ANSYS Mechanical APDL). The element nodes have six degrees of freedom each—UX, UY, UW, ROTX, ROTY, and ROTZ (three linear movements along and three rotation angles around the coordinate axes). Degrees of freedom UX, UY correspond to membrane, and UZ, ROTX, ROTY—bending deformations. The finite element meshes developed in the ANSYS Mechanical APDL and SCAD software packages are identical in terms of the number of nodes and elements that adequately reflect geometric, inertial, and load characteristics. Depending on the assumed gradient of change in the stress-strain state, the finite element mesh was divided into three zones. Finite element meshes broken down in the meridional direction with a step of 0.025 m at a height Z from the bottom, from 0.0 m to 2.5 m, with a step of 0.050 m at a height Z from the bottom with 2.5 m to 6.0 m in increments of 0.100 m at a height Z from the bottom 6.0 m to 9.5 m and in the circumferential direction in increments of 2.5º (144 element). This partitioning into finite elements allows us to refine the solution in the areas of high gradients of changes in the desired functions and is chosen based on the optimal calculation time and the accuracy of results based on preliminary test calculations.

When carrying out calculations, the allowable stress was taken for a given steel grade of 168 MPa. The value of the yield stress was taken to be 252 MPa, and the ultimate strength was 520 MPa. The value of the elastic modulus was taken to be 1.96-10^5 MPa and a Poisson’s ratio 0.29, and the density of steel 7.9 t/m^3. The material model is selected linearly elastic.

The boundary conditions are assigned by prohibiting all linear and angular displacements of all lower nodes of the model. Static boundary conditions are set hydrostatic pressure caused by the liquid filling density 1 t/m^3 based on the conditions of the assumed operating modes of the cylindrical tank.

2.3. Linear calculation results
The thickness selection was based on the third (maximum shear stress) and fourth (distortion energy) strength hypotheses, which are in good agreement with experimental data for plastic materials such as steel [10]. Based on preliminary linear calculation studies, the minimum required thicknesses for wall and partition elements were determined, which is defined as the greatest thickness when selecting according to two strength theories. The wall thickness is equal to \( t_w = 1.0 \) cm, and the partition thickness is \( t_p = 4.5 \) cm.
Let us present the results of linear calculation in two software packages for the most unfavorable operating mode (mode No. 3), shown in figure 3 and figure 4. As the parameters to be compared, consider the displacement (in the cartesian coordinate system), as well as the values of the largest equivalent (according to the fourth theory of strength) stresses.

**Figure 3.** Isofields of displacements $u_x$ in the Cartesian coordinate system according to the results of linear calculations in the ANSYS (left) and SCAD (right) software packages for mode No. 3, mm.

**Figure 4.** Isofields of equivalent stresses on the bottom layer of the shell based on the results of linear calculations in the ANSYS (left) and SCAD (right) software packages for mode No. 3, MPa.

As seen from figures 3 and 4 results of linear calculation in two software packages are well correlated with each other both in qualitative and quantitative characteristics.

Considering the results of the calculation during normal operation with the stress level not exceeding the maximum for the material, it is worth noting the most dangerous zones of the structure in terms of strength based on linear calculation. For the outer wall of a cylindrical tank, the maximum stresses are localized in the supporting part, however, it is worth noting the zones of high stresses along the height of the interface between the wall and the partition. For internal partitions, the greatest stresses are localized in the central zones of intersecting partitions, as well as in the interface with the outer wall, by analogy with a plate rigidly supported along the contour.
2.4. Nonlinear calculation results

To take into account the influence of membrane forces on bending effects, the interaction between bending effects and longitudinal forces (analogous to the phenomenon of transverse-longitudinal bending in beams) was considered in the form of geometric nonlinearity, when the work of an elastic system is associated with the need to take into account the change in the geometry of the system during its deformations under load. Geometric nonlinearity can be taken into account through equations relating displacements to deformations or using equilibrium equations. Calculations taking into account geometric nonlinearity are often referred to as deformed calculations.

In the SCAD software package, to take into account geometric nonlinearity, it is possible to take into account only the "Karman approximation", when it is assumed that the squares of the angles of rotation of the elements of the considered design scheme are quantities of the same order of smallness as the relative elongations in the material, which in turn are considered small in comparison with unit [11]. Nonlinear calculations are performed using a stepwise method, the idea of which is based on tracking the behavior of the system at relatively small load increments. In this case, at each step, a linearized system of resolving equations is solved for the current increment of the nodal load vector generated for the load under consideration. The calculation is focused on solving nonlinear problems in several modifications of the step method: a simple step method, a step method with refinements, a step-iterative method [12].

The ANSYS Mechanical APDL software package provides a Newton-Raphson approach to account for nonlinearities, in which the entire load is divided into a number of increments of this load. Before each solution, the Newton-Raphson method evaluates the unbalanced load vector, which is the difference between restoring forces (loads corresponding to element stresses) and applied loads. The program then performs a linear solution using unbalanced loads and verifies convergence. If the convergence criteria are not met, the unbalanced load vector is reevaluated, the stiffness matrix is recalculated and a new solution is obtained. This iterative procedure continues until the problem reaches convergence according to user-specified criteria [13].

Based on the thicknesses obtained from the linear calculation, a refined nonlinear numerical study of the stress-strain state was carried out taking into account the membrane forces on the bending component, the results of which are shown in figure 5 and figure 6.

![Figure 5](image_url)  ![Figure 6](image_url)

**Figure 5.** Isofields of displacements $u_x$ in the Cartesian coordinate system according to the results of nonlinear calculations in the ANSYS (left) and SCAD (right) software packages for mode No. 3, mm.

As seen from figure 5 and 6, the results of nonlinear calculation in two software packages are well correlated with each other qualitatively and quantitatively.

In comparison with the results of linear calculations, the overall stress level slightly decreased, however, in the local areas of intersection of elements with each other and in the zone of the support
part, the stress state underwent only a slight change, since here the stresses depend little on geometric nonlinearity.

![Isofields of equivalent stresses](image)

**Figure 6.** Isofields of equivalent stresses on the inner layer of the shell according to the results of nonlinear calculations in software packages ANSYS (left) and SCAD (right) for mode No. 3, MPa.

2.5. *Comparison of calculation results*

Comparative analysis of computational studies using alternative software packages is an integral part of the computational process associated with the verification of a numerical solution.

Comparative analysis of computational studies (table 1) presents the results of calculations in software packages with each other and a comparison of solutions in various settings (linear and nonlinear).

| Calculation type       | Compared parameters | ANSYS Mechanical APDL | SCAD | Δ, % |
|-----------------------|---------------------|------------------------|------|------|
| Linear solution       | $u_{\text{max}}$, mm | 16.85                  | 16.90| 0.30 |
|                       | $\sigma_{\text{max}}$, MPa | 153.47                  | 169.83| 10.66 |
| Nonlinear solution    | Method              | Newton-Raphson         | Step       |
|                       | $u_{\text{max}}$, mm | 13.63                  | 13.73| 0.73 |
|                       | $\sigma_{\text{max}}$, MPa | 137.14                  | 159.58| 16.36 |
| δ, %                  | $\delta_u$, %       | 23.62                  | 23.01|
|                       | $\delta_{\sigma}$, % | 11.91                  | 6.41 |

3. **Conclusions**

As a result of the performed computational studies of the stress-strain state of an atypical construction of a vertical four-compartment cylindrical tank with inner partitions for storing liquid under various operating modes, the following conclusions and recommendations can be formulated:

1. The finite element models of the supporting structures of the atypical vertical four-compartment cylindrical tank were developed in the software packages ANSYS Mechanical APDL and SCAD. The results of the comparative analysis show similar resultant stress-strain states in both linear and nonlinear formulations.

2. Based on the performed computational studies in linear and nonlinear formulations, it was determined that taking into account the membrane forces on the bending component of the considered atypical structure does not change the qualitative characteristics, but changes the quantitative characteristics of the stress-strain state: the difference in the maximum displacements $\delta_u$ is about 20% to a lesser side, the difference in the maximum stresses $\delta_{\sigma}$ of the order of 6-12% is also downward. This fact indicates that when calculating such structures, it is allowed to use a linear relationship between
forces and displacements, since such a formulation of the calculation goes into a margin of safety. However, it should be noted that this conclusion is valid for the considered tank design. With an increase in the size of the tank, as well as the spans of the partitions inside, the geometric nonlinearity can make a greater contribution to the stress-strain state.

3. Based on the calculated studies, it was found that the use of a continuous steel sheet as a partition of the structure is impractical, because according to the results of calculations, the thickness of the steel sheets turned out to be significant (up to 4.5 cm).

4. Calculations in a geometrically nonlinear formulation, taking into account the influence of membrane forces on bending effects in order to justify a decrease in pre-assigned thicknesses from a linear calculation, does not lead to a significant change in the stress-strain state and reasoning to a decrease in the thicknesses of steel sheets for the considered construction, since the greatest stresses are concentrated in local areas of interface shells and are less sensitive to geometric nonlinearity.

5. The considered disadvantages allow us to conclude that this structural scheme of the tank with continuous steel partitions is impractical due to their significant thicknesses, and therefore requires the development of a design diagram of the partition, which provides mechanical safety and tightness in conjunction with economic indicators and technological features of a construction.

References
[1] Shagivaleev K F and Burov S N 2011 Calculation of thin-walled spatial systems consisting of interconnected closed cylindrical shells using the force method Building mechanics of engineering structures and structures I pp 8–12
[2] Kositsyn S B and Tran Xuan Linh 2013 Analysis of the stress-strain state of intersecting cylindrical shells under elastic-plastic deformations taking into account geometric nonlinearity Structural mechanics of engineering structures and structures. Review-analytical and scientific-technical journal (Moscow) I pp 3–9
[3] Kositsyn S B and Tran Xuan Linh 2014 Comparative analysis of various models of material as applied to the calculations of the shell of a railway tank taking into account elastoplastic deformations and geometric nonlinearity Mechanization of construction 838 (4) pp 12–5
[4] Kositsyn S B and Tran Xuan Linh 2014 Numerical analysis of the stress-strain state of orthogonally intersecting cylindrical shells without and taking into account their one-way interaction with the surrounding soil mass International Journal for Computational Civil and Structural Engineering 10 (1) pp 72–8
[5] Manuilov G A and Kositsyn S B 2006 Investigation of the loss of stability of the railway tank shell Computational mechanics of a deformable solid - Proceedings of the international scientific and technical conference. In two volumes (Moscow: Russian University of transport) pp 236–240
[6] Sherstobitova R T 2008 Improving the safety of vertical steel tanks operated for a long time at low temperatures (Ufa: Ufa State Petroleum Technological University) p 131
[7] Rozin L A 2000 Finite element method Soros Educational Journal 6 (4) pp 120 –7
[8] Buryaka V A, Fokin V G, Soldusova E A, Glazunova N A and Adeyanov I E 2010 Engineering Analysis in ANSYS Worckbench Textbook manual. SamSTU p 271
[9] Chigarev A V, Kravchuk A S and Smalyuk A F 2004 ANSYS for engineers Mechanical Engineering (Moscow) p 511
[10] Vardanian G S, Andreev V I, Atarov N M and Gorshkov A A 2011 Strength of materials with the fundamentals of theory of elasticity and plasticity ASV(Moscow) p 568
[11] Perelmuter A V and Sliwker V I 2007 Computational models of structures and the possibility of their analysis DMK Press (Moscow) p 600
[12] Karpilovsky V S, Kriksunov E Z, Malyarenko A A, Mikitarenko M A, Perelmuter A V and Perelmuter M A 2015 SCAD Office. Computing complex SCAD ++ ASV (Moscow) p 848
[13] ANSYS, Inc 2018 Structural Analysis Guide. Release 19.2