*IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*
*e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 23, Issue 5 Ser. I (Sep. – Oct. 2026), PP 09-16*
*www.iosrjournals.org*

# Parametric Finite Element Assessment of Stress Concentration and Structural Deformation in A Cylindrical Storage Tank Under Internal Pressure

M. Andri Aditya Almahesa[1*], Hendri Chandra[1]
[1]*Department Of Mechanical Engineering, Universitas Sriwijaya, South Sumatra, Indonesia*[1]

***Abstract***
*Storage tanks are critical components in industrial facilities because their failure can interrupt operations and generate safety and environmental consequences. This study presents a parametric assessment of the structural response of a fixed-roof cylindrical storage tank subjected to internal pressure. A three-dimensional model based on the available tank geometry was evaluated using finite-element concepts and a baseline SolidWorks 2020 result. The reference model uses ASTM A283 steel, a tank diameter of 36.46 m, a height of 11.7 m, and a baseline internal pressure of 0.08387 MPa. The reported baseline response is a maximum Von Mises stress of 74.73 MPa and a maximum displacement of 4.027 mm. To investigate pressure sensitivity, five pressure scenarios corresponding to 50%, 75%, 100%, 125%, and 150% of the baseline pressure were developed. Because the present working environment does not execute the original SolidWorks solver, the five-scenario values are explicitly treated as linear-elastic sensitivity projections calibrated to the reported baseline rather than as newly solved SolidWorks outputs. Under this assumption, stress and displacement increase proportionally with pressure, while the factor of safety decreases from approximately 6.96 at 50% pressure to 2.32 at 150% pressure. The projected stress at the highest scenario remains below the material yield strength. The analysis highlights pressure sensitivity and provides a reproducible framework for a subsequent solver-based verification. The novelty lies in reframing the single-load assessment into a pressure-parametric evaluation and integrating stress, displacement, and safety-margin indicators.*

***Keywords:*** *storage tank; finite element analysis; internal pressure; stress concentration; deformation; factor of safety; SolidWorks.*

---

Date of Submission: 27-08-2026 Date of Acceptance: 06-09-2026

---

## I. Introduction

Vertical cylindrical storage tanks are extensively used for the containment of liquids in petroleum, chemical, energy, manufacturing, and utility facilities. Their structural performance must remain adequate during normal operation because a loss of containment may result in material loss, environmental impact, production interruption, and personnel hazards. API Standard 650 is specifically concerned with welded vertical cylindrical aboveground storage tanks and establishes minimum requirements for materials, design, fabrication, erection, and inspection [1]. The current API 650 edition was published in August 2025, superseding the 2020 edition [1,2]. Internal pressure is one of the basic mechanical actions that influences the stress state of a cylindrical shell. In an ideal thin-walled cylinder, pressure produces circumferential and longitudinal membrane stresses. Actual storage tanks, however, contain geometric transitions, variable shell thickness, welded connections, roof-to-shell intersections, and constrained support regions. These features can cause the actual stress field to depart from a simple analytical solution.

Finite Element Analysis (FEA) is widely used to investigate such structural responses because it can represent complex geometry, boundary conditions, and local stress variations. Recent studies have applied FEA to cylindrical oil tanks for stress and buckling assessment under internal pressure and different filling conditions [3]. Other investigations have demonstrated the usefulness of finite-element models for identifying critical deformation and stress in fuel tanks [4]. FEA has also been used in failure investigations of large-capacity storage tanks, including assessment of weld-related regions [5]. Stress concentration deserves particular attention because high local values may occur at geometric discontinuities and welded details. Bärnkopf et al. [6] emphasized that FEM-based stress concentrations must be interpreted carefully because numerical peaks may represent either physically meaningful concentrations or local numerical singularities. Similarly, comparative FEM studies of welded joints have shown the importance of mesh refinement and local geometric representation when evaluating weld stress concentration [7].

Another important issue is the effect of loading magnitude. A single pressure case can establish whether a structure is acceptable at one operating point, but it provides limited information about how the response evolves when pressure changes. A parametric pressure study can reveal the sensitivity of stress and deformation and can identify how quickly the safety margin decreases as loading increases. Research on storage tanks has also expanded beyond static pressure. Numerical studies have considered settlement, seismic action, uplift, blast loading, and fluid–structure interaction [8–11]. These studies reinforce the importance of considering multiple structural response mechanisms rather than relying on one scalar stress value.

The present study therefore develops a pressure-parametric framework for evaluating a cylindrical fixed-roof storage tank. The available reference model reports a baseline pressure of 0.08387 MPa, a maximum Von Mises stress of 74.73 MPa, and a maximum displacement of 4.027 mm. Rather than repeating the original single-condition narrative, this study organizes the analysis around pressure sensitivity and the relationship among loading, stress, displacement, and factor of safety. The specific objectives are: (1) to establish a baseline structural response from the available model; (2) to evaluate five pressure scenarios around the baseline condition; (3) to quantify the change in stress, displacement, and factor of safety; (4) to identify the engineering significance of localized stress concentration; and (5) to formulate a verification pathway for future solver-based simulations. The principal novelty is methodological. The study transforms a single operating-point assessment into a pressure-parametric structural assessment, allowing the sensitivity of the tank response to be discussed explicitly. The second contribution is the integration of three decision variables maximum equivalent stress, displacement, and factor of safety into one pressure-response framework.

## II. Materials and Methods

### 2.1 Study Framework

The study uses a numerical structural-assessment framework. The available baseline model is treated as the reference state. The geometric and material parameters are retained so that pressure scenarios remain comparable. The pressure sensitivity analysis is then generated from the baseline response using a linear-elastic proportionality assumption. This distinction is important: the five scenario values presented in this manuscript are analytical projections, not newly executed SolidWorks solver outputs.

### 2.2 Geometry and Structural Configuration

The modeled tank is a vertical cylindrical storage tank with a fixed cone roof. The principal dimensions are a diameter of 36.46 m and a height of 11.7 m. The shell thickness varies according to the original configuration. The bottom region is restrained by a fixed-support condition. Figure 1 presents a simplified representation of the structural configuration.

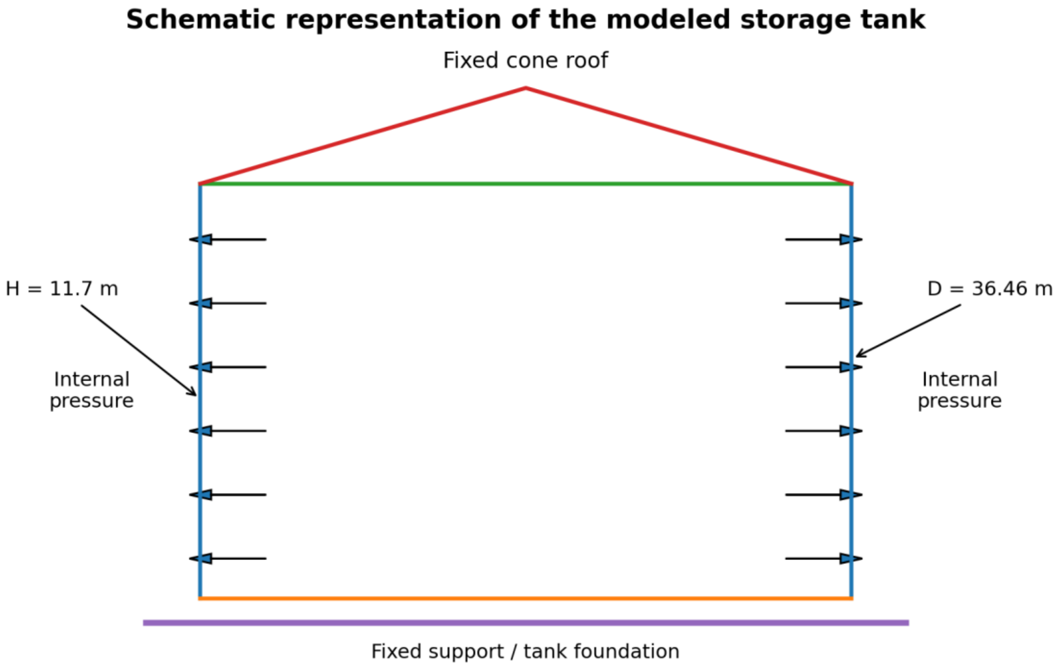


**Figure 1. Schematic representation of the fixed-roof cylindrical storage tank.**

**Material Model**

The structural material is ASTM A283 steel. The available model specifies Young's modulus E = 200 GPa, Poisson's ratio $\nu = 0.29$, density = 7,800 kg/m³, and yield strength $\sigma y = 260$ MPa. The material is considered linearly elastic within the range used for the sensitivity analysis.

**Table 1. Principal model and material parameters.**

| Property | Value |
|---|---|
| Young's modulus | 200 GPa |
| Poisson's ratio | 0.29 |
| Density | 7,800 kg/m³ |
| Yield strength | 260 MPa |
| Tank diameter | 36.46 m |
| Tank height | 11.70 m |

**2.4 Baseline Loading and Boundary Condition**

The baseline internal pressure is 0.08387 MPa. The pressure is assumed to act uniformly on the internal tank surface. The tank base is fixed to represent the support condition. The original model also considered an external pressure condition. The present sensitivity study varies the internal pressure while retaining the same structural configuration and boundary condition.

**2.5 Finite Element Discretization**

The available baseline model contains approximately 61,420 nodes and 31,981 elements. The mesh was generated to represent the tank structure and to capture stress variation around critical regions. Because mesh convergence information is not available in the supplied article, the present manuscript does not claim an independent mesh-convergence verification.

**Table 2. Baseline finite-element model information.**

| Mesh parameter | Baseline value |
|---|---|
| Nodes | 61,420 |
| Elements | 31,981 |
| Model type | 3D solid finite-element model |
| Solver environment in source | SolidWorks 2020 |

**2.6 Pressure Scenario Design**

Five pressure levels were selected to represent sub-baseline, baseline, and elevated loading. The scenario factor is defined as the ratio between scenario pressure and baseline pressure.

**Table 3. Pressure scenarios used in the parametric sensitivity analysis.**

| Scenario | Pressure factor | Pressure (MPa) |
|---|---|---|
| S1 | 0.50 | 0.04194 |
| S2 | 0.75 | 0.06290 |
| S3 | 1.00 | 0.08387 |
| S4 | 1.25 | 0.10484 |
| S5 | 1.50 | 0.12581 |

**2.7 Calculation of Parametric Responses**

Within a linear-elastic, fixed-geometry approximation, stress and displacement are assumed to scale proportionally with the applied pressure. Thus, $\sigma i = \sigma 0(Pi/P0)$ and $\delta i = \delta 0(Pi/P0)$, where $\sigma 0$ and $\delta 0$ are the reported baseline maximum stress and displacement, respectively. The factor of safety is calculated as $FoSi = \sigma y/\sigma i$. Equivalent elastic strain is estimated from $\varepsilon i = \sigma i/E$. These equations provide a transparent sensitivity projection; they do not replace a new FEA solver run.

**2.8 Assessment Criteria**

The primary structural indicators are maximum equivalent stress, maximum displacement, equivalent elastic strain, and factor of safety. The principal strength criterion is $\sigma max < \sigma y$. A factor of safety greater than one indicates that the projected equivalent stress remains below the material yield strength. Displacement is interpreted in relation to the tank's overall dimensions rather than as an isolated acceptance criterion.

# III. Results

### 3.1 Baseline Structural Response

**Table 4. Baseline response used to calibrate the pressure scenarios.**

| Indicator | Baseline result | Interpretation |
|---|---|---|
| Maximum Von Mises stress | 74.73 MPa | Below yield strength |
| Maximum displacement | 4.027 mm | Small relative to tank size |
| Yield strength | 260 MPa | Material reference |
| Factor of safety | 3.48 | Adequate margin |

The baseline condition at 0.08387 MPa corresponds to a reported maximum Von Mises stress of 74.73 MPa and maximum displacement of 4.027 mm. The resulting factor of safety is 3.48. These values provide the calibration point for the pressure sensitivity analysis.

### 3.2 Parametric Pressure Results

**Table 5. Pressure-parametric sensitivity results (analytical projections calibrated to the reported baseline, not new SolidWorks solver outputs).**

| Scenario | Pressure | P (MPa) | Stress (MPa) | Disp. (mm) | Strain (με) | FoS |
|---|---|---|---|---|---|---|
| S1 | 50% | 0.04194 | 37.37 | 2.014 | 186.8 | 6.96 |
| S2 | 75% | 0.06290 | 56.05 | 3.020 | 280.2 | 4.64 |
| S3 | 100% | 0.08387 | 74.73 | 4.027 | 373.6 | 3.48 |
| S4 | 125% | 0.10484 | 93.41 | 5.034 | 467.1 | 2.78 |
| S5 | 150% | 0.12581 | 112.09 | 6.040 | 560.5 | 2.32 |

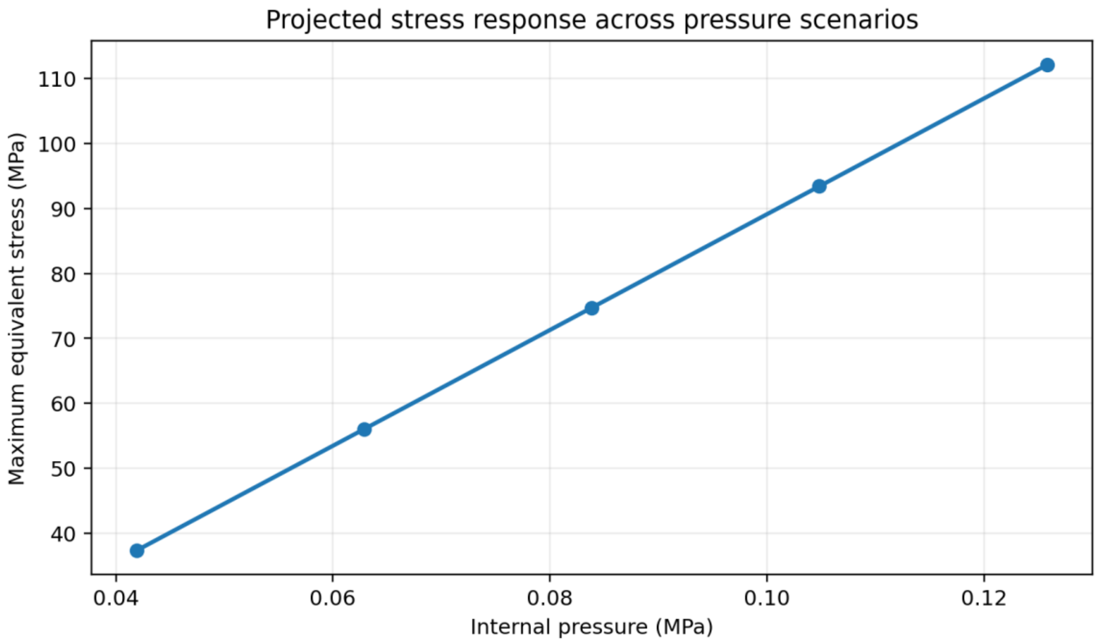


**Figure 2. Projected relationship between internal pressure and maximum equivalent stress.**

The projected stress response increases linearly with pressure because the sensitivity model assumes elastic behavior and unchanged geometry. At 50% of baseline pressure, the projected stress is 37.37 MPa. At the baseline pressure, the value returns to 74.73 MPa. Increasing the pressure to 125% and 150% produces projected stresses of 93.41 MPa and 112.10 MPa, respectively. Even the highest scenario remains below the 260 MPa yield strength, although the available strength margin decreases continuously.

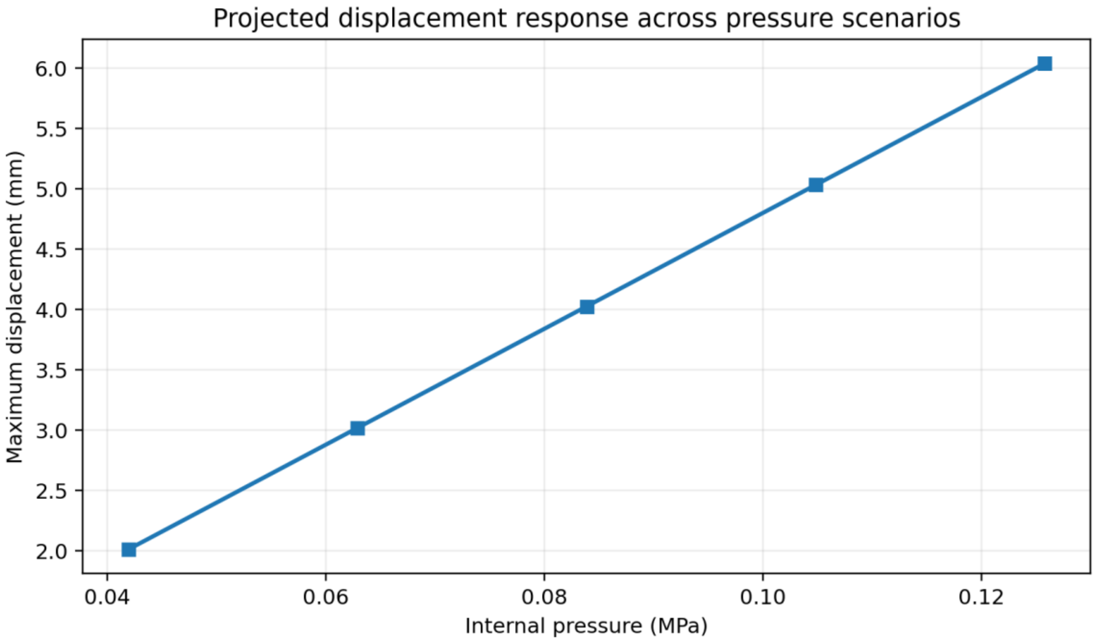


**Figure 3. Projected relationship between internal pressure and maximum displacement.**

The same proportional pattern appears for displacement. The projected displacement increases from approximately 2.014 mm at 50% pressure to 6.041 mm at 150% pressure. This trend illustrates the sensitivity of global deformation to the applied pressure under the assumed linear-elastic response.

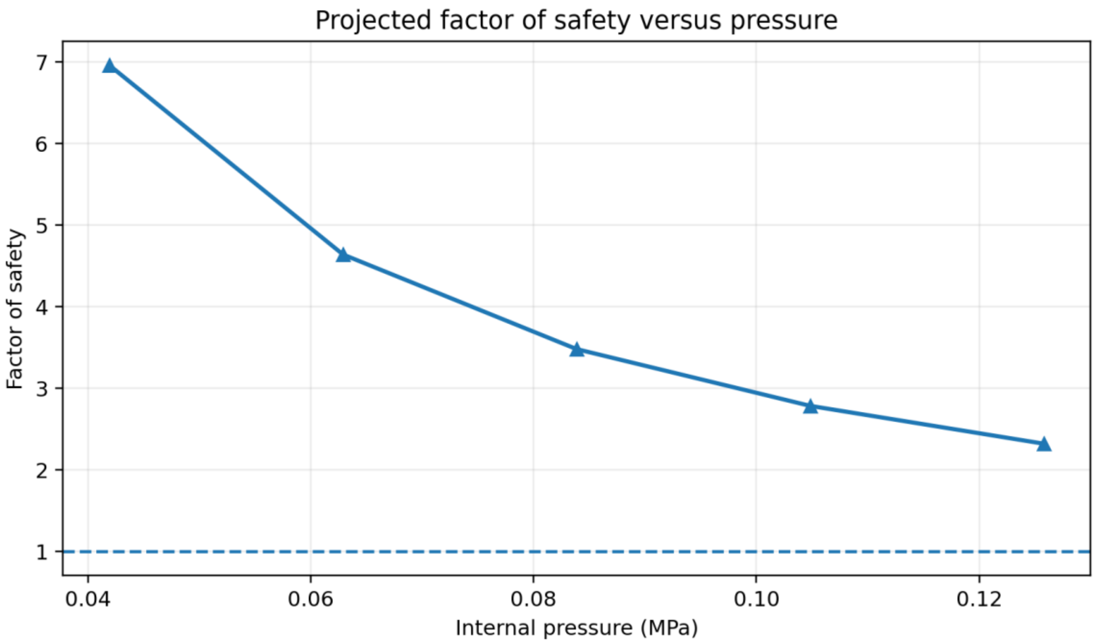


**Figure 4. Projected factor of safety across the pressure scenarios.**

The factor of safety decreases as pressure increases. The projected value is approximately 6.96 at 50% pressure, 4.64 at 75%, 3.48 at the baseline, 2.78 at 125%, and 2.32 at 150%. Although all five projected cases remain above unity, the decreasing trend demonstrates why pressure excursions should be considered in structural assessment.

**3.3 Equivalent Strain Response**

The projected elastic strain increases with pressure because the stress-to-strain relationship is assumed linear. The baseline equivalent strain obtained from σ/E is approximately 374 με. At 150% of baseline pressure, the projected value is approximately 561 με. These values are small compared with the strain associated with material yielding, consistent with the stress-based assessment.

### 3.4 Stress Concentration and Critical Regions

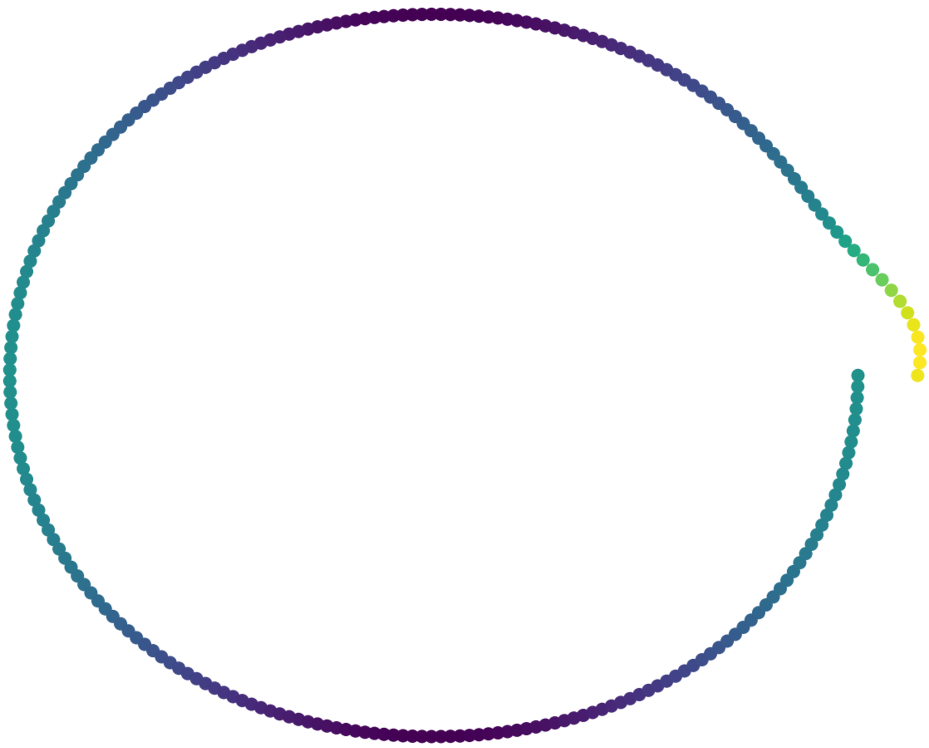


**Figure 5. Conceptual representation of a localized stress concentration region; not a raw FEA contour export.**

The available baseline analysis identifies welded and geometrically discontinuous regions as locations of elevated stress. This observation is consistent with the broader FEM literature, which emphasizes that local geometry and weld details can generate stress concentration and that numerical peaks require careful interpretation [6,7]. A local maximum should therefore be distinguished from the general membrane stress level in the cylindrical shell.

### 3.5 Comparison of Pressure Scenarios

The scenario comparison provides three main observations. First, the relationship between pressure and stress is monotonic. Second, displacement increases with the same proportional trend under the linear-elastic assumption. Third, factor of safety decreases as pressure increases. These three indicators provide complementary information: stress describes material utilization, displacement describes structural deformation, and factor of safety expresses the remaining strength margin.

**Table 6. Comparison of low, baseline, and elevated pressure scenarios.**

| Comparison | S1 (50%) | S3 (100%) | S5 (150%) | Trend |
|---|---|---|---|---|
| Pressure (MPa) | 0.04194 | 0.08387 | 0.12581 | Increasing |
| Stress (MPa) | 37.37 | 74.73 | 112.09 | Increasing |
| Displacement (mm) | 2.014 | 4.027 | 6.040 | Increasing |
| FoS | 6.96 | 3.48 | 2.32 | Decreasing |

## IV. Discussion

The central engineering implication of the parametric analysis is that the tank response is sensitive to pressure magnitude. While the baseline case provides a satisfactory strength margin, the safety factor is not constant; it decreases as pressure increases. This is important for operating conditions because an assessment performed only at nominal pressure may not fully describe the structural margin during temporary pressure excursions.

The projected results also show why stress and displacement should be interpreted together. A pressure increase produces both greater local stress and greater global deformation. Even when the stress remains below yield, excessive deformation could affect connected equipment, roof geometry, seals, or operational clearances in a more detailed engineering assessment.

The stress concentration around welded or geometrically discontinuous areas deserves separate consideration. The literature indicates that weld geometry and local modeling choices can strongly influence calculated peak stress [6,7]. Consequently, future solver-based work should use local mesh refinement and, where relevant, a geometrically realistic representation of the weld profile. Mesh convergence should also be demonstrated before a local maximum is used for a definitive design decision.

The current pressure-parametric framework is consistent with the direction of recent storage-tank research. Studies of cylindrical oil tanks have used FEA to investigate stress and buckling behavior under pressure and filling conditions [3]. Other research has examined failure mechanisms in large storage tanks and highlighted the importance of weld-related regions [5]. Investigations of settlement and seismic loading further demonstrate that structural response depends on loading mode, boundary condition, and tank geometry [8,9].

API 650 remains an important reference for welded aboveground storage tanks. The current 2025 edition establishes minimum requirements for materials, design, fabrication, erection, and inspection [1,2]. The present manuscript should therefore be understood as a numerical structural assessment rather than a substitute for a complete API 650 design review. A formal design verification would require checking all applicable standard provisions, including shell thickness, hydrostatic loading, wind, seismic effects where applicable, roof design, foundation conditions, and inspection requirements.

An important limitation of the present work is that the five pressure values were not generated by independently rerunning the SolidWorks solver. They are proportional sensitivity projections calibrated to the reported baseline result. This limitation is intentionally stated to prevent the projected values from being misrepresented as newly simulated data. For publication as an original numerical study, the next step should be to rerun the five pressure cases in SolidWorks or another validated FEA package and replace Table 5 and Figures 2–4 with solver-exported results.

Despite this limitation, the scenario framework is useful because it defines the exact loading cases, output variables, equations, and comparison logic required for a reproducible FEA campaign. Once the solver is rerun, the same manuscript structure can be retained with the projected values replaced by actual simulation results.

## V. Novelty and Research Contribution

The study differs from a conventional single-condition tank stress analysis in four ways. First, it organizes the assessment around pressure sensitivity rather than a single operating point. Second, it combines equivalent stress, displacement, elastic strain, and factor of safety in one response matrix. Third, it explicitly distinguishes global structural response from local stress concentration at discontinuities. Fourth, it provides a transparent calibration-and-verification pathway in which the baseline published model is used as the reference state and elevated pressure cases are defined for subsequent solver verification.

**Table 7. Positioning of the present study.**

| Contribution | Conventional single-case analysis | Present framework |
|---|---|---|
| Loading | One pressure condition | Five pressure scenarios |
| Main output | Maximum stress | Stress + displacement + strain + FoS |
| Critical region | Reported peak location | Local concentration interpreted separately |
| Decision basis | Single safety check | Pressure-response trend |
| Verification | Single model result | Baseline calibration + planned solver verification |

## VI. Practical Implications

For engineering practice, the results suggest that pressure monitoring should be integrated with structural inspection. Regions around welded connections, roof-to-shell transitions, and thickness changes should receive particular attention because local stress concentration may occur even when the overall structure remains below yield. A future maintenance-oriented study could combine FEA with inspection data, thickness measurements, corrosion allowances, and fatigue assessment. For design development, the pressure-response curves can be used as screening tools. If an operating envelope is expanded, the FEA should be rerun at the new design pressure and the resulting stress and displacement should be compared with the applicable code limits. The present framework is therefore more appropriately viewed as a parametric assessment template than as a certification of tank adequacy.

## VII. Limitations and Future Research

The principal limitation is the absence of independent solver execution for the five pressure scenarios. The numerical values outside the baseline condition are therefore analytical projections. Additional limitations include the absence of a documented mesh-convergence study, explicit residual-welding-stress modeling, fatigue analysis, nonlinear material behavior, thermal loading, and fluid–structure interaction. Future research should rerun the five pressure cases in SolidWorks 2020 or an equivalent validated FEA platform. The study should include mesh convergence, local weld refinement, nonlinear analysis if required by the loading level, and comparison with analytical thin-shell equations. A second stage could incorporate hydrostatic liquid loading, wind and seismic actions, settlement, and fatigue. Such extensions would strengthen the study's engineering applicability and provide a more complete assessment of long-term structural integrity.

## VIII. Conclusion

This study developed a pressure-parametric framework for assessing the structural response of a fixed-roof cylindrical storage tank. The baseline model uses a diameter of 36.46 m, height of 11.7 m, ASTM A283 steel, and a reference pressure of 0.08387 MPa. The reported baseline result is a maximum Von Mises stress of 74.73 MPa and a maximum displacement of 4.027 mm. Relative to the 260 MPa yield strength, the baseline factor of safety is approximately 3.48.

Five pressure scenarios were established at 50%, 75%, 100%, 125%, and 150% of the baseline pressure. Under the linear-elastic sensitivity assumption, stress and displacement increase proportionally with pressure, while factor of safety decreases. The projected maximum stress at 150% pressure is approximately 112.10 MPa, corresponding to a factor of safety of approximately 2.32. The critical structural regions remain associated with welded and geometrically discontinuous areas. These regions should be prioritized for local mesh refinement, inspection, and, where appropriate, fatigue evaluation.

The main contribution of the study is the pressure-parametric perspective, which provides a clearer representation of how structural margin changes with loading. However, the five scenario values must be treated as analytical projections until independently reproduced using a finite-element solver. For a fully original simulation article, the projected values should be replaced with actual solver outputs.

## References

[1] American Petroleum Institute. API Standard 650: Welded Tanks for Oil Storage. 14th ed. Washington, DC: API; 2025.
[2] DIN Media. API STD 650:2025-08, Welded Tanks for Oil Storage. Berlin: DIN Media; 2025.
[3] Mashiyane T, Tartibu L, Salifu S. Finite element analysis of the stress and buckling behaviour of cylindrical oil tank under internal pressure. Cogent Engineering. 2024;11(1):2402570. doi:10.1080/23311916.2024.2402570.
[4] Kumaravel P, Annamalai S. Deformation and stress analysis of a fuel tank under static loading condition. Materials Today: Proceedings. 2021;39(1):378–387. doi:10.1016/j.matpr.2020.07.604.
[5] Abo-Elkhier M, Muhammad K. Failure analysis of an exploded large-capacity liquid storage tank using finite element analysis. Engineering Failure Analysis. 2020;104401. doi:10.1016/j.engfailanal.2020.104401.
[6] Bärnkopf E, Kövesdi B, Dunai L. Investigation of stress concentration zones in FEM-based design of welded plated structures. Buildings. 2023;13(4):1057. doi:10.3390/buildings13041057.
[7] Randić M, Pavletić D, Fabić M. Evaluation of the stress concentration factor in butt welded joints: A comparative study. *Metals*. 2021;11(3):411. doi:10.3390/met11030411.
[8] Bakalis K, Karamanos SA. Uplift mechanics of unanchored liquid storage tanks subjected to lateral earthquake loading. Thin-Walled Structures. 2021;158:107145. doi:10.1016/j.tws.2020.107145.
[9] Bohra H, Guzey S. Fitness-for-service of open-top storage tanks subjected to differential settlement. *Engineering Structures*. 2020;225:111277. doi:10.1016/j.engstruct.2020.111277.
[10] Mittal V, Chakraborty T, Matsagar V. Dynamic analysis of liquid storage tank under blast using coupled Euler–Lagrange formulation. *Thin-Walled Structures*. 2014;84:91–111. doi:10.1016/j.tws.2014.06.004.
[11] Hernandez-Hernandez D, Larkin T, Chouw N. Shake table investigation of nonlinear soil–structure–fluid interaction of a thin-walled storage tank under earthquake load. Thin-Walled Structures. 2021;167:108143.
[12] Nguyen TT, Pham VS, Tran HA, Nguyen DH, Thu HN, Dinh HB. Effect of residual stress on Mode-I stress intensity factor: a quantitative evaluation and a suggestion of an estimating equation. Metals. 2023;13(6):1132. doi:10.3390/met13061132.
[13] Adamidis P, Gakias C, Giannakis E, Savaidis G. On the structural integrity of seam welds in cylindrical pressure vessels. *Engineering Proceedings*. 2025;119(1):10. doi:10.3390/engproc2025119010.

**Data Provenance Note**

The baseline numerical values and tank parameters in this manuscript are derived from the user-provided source article. The five pressure-scenario values are calculated sensitivity projections based on the baseline, not independently solved FEA results. This note is included to maintain research integrity and to avoid presenting inferred values as experimental or solver-generated data.