International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Role of Finite Element Methods (FEM) in Pressure Vessel Nozzle Stress Analysis: A Survey of Applications and Trends

Vikas Thakran*

Independent Researcher

Received 01 Dec 2024, Accepted 20 Dec 2024, Available online 23 Dec 2024, **Vol.14, No.6 (Nov/Dec 2024)**

Abstract

Finite Element Methods (FEM) have emerged as a cornerstone in the structural analysis of pressure vessel nozzles, addressing critical challenges like stress concentrations, material nonlinearities, and thermal loads. This paper surveys FEM applications in nozzle stress analysis, comparing 2D and 3D models for efficiency and accuracy, analyzing complex geometries and multi-nozzle systems, and ingrating material behavior under combined mechanical and thermal stresses. By leveraging FEM tools such as ANSYS and ABAQUS, engineers can model intricate configurations, refine designs, and ensure safety and reliability in industrial applications. Future advancements, including AI-assisted FEM, hold promise for further enhancing precision and efficiency in structural analysis.

Keywords: *Finite Element Methods (FEM), Pressure Vessel Nozzles, Stress Analysis, 2D and 3D FEM Models, Material Nonlinearities, Thermal Loads, Stress Concentrations, Multi-Nozzle Systems, AI-Assisted FEM, Structural Analysis, Design Optimization.*

1. Introduction

A pressure vessel is a closed space where pressure may be applied from either the inside or the outside. They are most often utilized in industry as storage tanks, reactors, heat exchangers, etc. The confined fluid may leak out as a result of pressure action, which might result in a major accident and fatalities. Consequently, a few regulatory bodies, such as ASME, BS, and API standards, among others, regulate the design, manufacture, and testing methods. A certification from one of the legislatures is required for all pressure vessels utilized in the industry. Pressure vessels with varying shell thicknesses have undergone thermal stress study. A pressure of 1.034 MPa is exerted on the inside surface of the pressure vessel, which is supported by the saddles. Finding the deformation and thermal stress that has developed on a pressure vessel is the goal of thermal stress analysis [1].

Analyzing the stress on nozzles that are welded to pressure vessels is a very important part of designing these vessels. Over time, many techniques have been developed to deal with the main problems caused by stress build-up, how materials interact, and forces from outside. These methods have been widely used for analyzing stress in nozzles, providing useful information about cylindrical shell nozzles and various loading situations.

*Corresponding author's ORCID ID: 0000-0000-0000-0000
DOI: https://doi.org/10.14741/ijcet/v.14.6.13

However, these methods have some drawbacks when it comes to accurately measuring stress concentrations where nozzles meet cylinders, especially under complex thermal and mechanical loads.

To overcome these challenges, Finite Element Analysis (FEA) has moved a step ahead of the conventional method as a better way of evaluating nozzle stresses. Previous works have compared in detail WRC methods and FEA, pointing out their advantages and drawbacks, and underlining how both the techniques are complementary, and should therefore be used in combination to produce highly accurate stress analysis [2]. This has been constant inspection and repair on the various in-service pressure vessel nozzles serves to underpin the rationale for the discovery of better stress analysis technology to prevent middle-of-life failures and promote safety [3][4].

Take a moment and imagine the world of structural analysis without computational techniques. Structural design graphs and models were bound for structural analysis, but these other methods could never achieve accuracy and precision. The Structural Analysis monolith industry would have to wait until the development of mid-20th century when engineers scaled FEM down. Coupled together with tools like ABAQUS and ANSYS, intricate load structures saw great advancements. In the last few years, combination of AI and its subset called machine learning has introduced new possibilities for modeling and optimization in structures. It not only solves the problem such as stress concentration and material nonlinearity but also provides real-time monitoring and adaptive analysis so that more safety and reliability is provided to the engineering structures [5].

Structure of the Paper

The structure of this paper is as follows: Section II provides a basics of pressure vessel nozzle stress analysis. Section III discusses overview of finite element methods (FEM). Section IV addresses FEM applications in pressure vessel nozzle stress analysis. Section V reviews relevant literature, and Section VI concludes with key findings and future research directions.

Basics of pressure vessel nozzle stress analysis

Pressure vessel nozzles play a critical role in facilitating fluid flow, serving as connection points for pipelines, valves, and other equipment. Their structural integrity is vital to the safety and performance of the entire vessel. The stresses encountered in pressure vessels, particularly around nozzles, include hoop stress, radial stress, and localized stress concentrations, which can arise due to internal pressure, external loads, and thermal effects [6]. Conventional methods for stress analysis, such as the use of simplified empirical formulas and analytical approaches, have historically been employed to estimate these stresses.

Role of Nozzles in Pressure Vessels

In any pressure vessel, the nozzle is part of the component providing the important functions of serving outlets, inlets, drains, vents, and manholes. These are the weld-in connections for a pressure vessel to an external piping system and may be situated either on the cylindrical shell or on the two ends of the pressure vessel [7]. For the identification of stress concentrations around the openings of nozzles in the pressure vessel, nozzles have functional consequence. Furthermore, a number of extra design problems arise due to new end-connected nozzles and eccentricity in the connection-nozzle placement as eccentric nozzles contribute to the warping of the structure [8].

Types of Stresses Encountered

There are two types of stress encountered discuss below:

Radial Stress: The directions of radial stress are those that are perpendicular to the axis of symmetry and also coplanar with it. On the interior of a thick-walled pipe, the radial stress is zero illustrate in Figure 1, whereas on the outside, it is equal to and opposite to the gauge pressure. Compressive stress is always present in radial stress. For a pressure vessel, radial stress is calculated as in Equation (1):

$$\sigma_r = -P \quad (1)$$

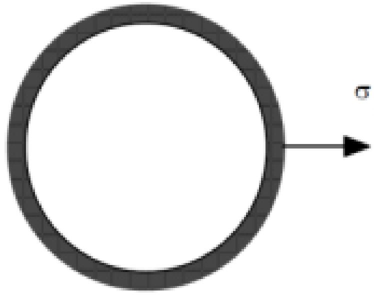

Fig.1 Radial Stresses in a Pressure Vessel

On the inside of the tank and its nozzle, the radial tension is equal to the pressure within, −P, while it becomes negligible at the outer surface.

Hoop stress: A pressure vessel is prone to splitting in half when subjected to hoop stress depict in Figure 2. It is indeed feasible for the vessel to fail in two halves along any axis and radius. The hoop stress is often expressed in Equation (2) as follows for a cylindrical pressure vessel operating at an internal pressure of P:

$$\sigma h = P \cdot r/t \quad (2)$$

Where: P: Internal pressure, r: Radius of the vessel and t: Wall thickness

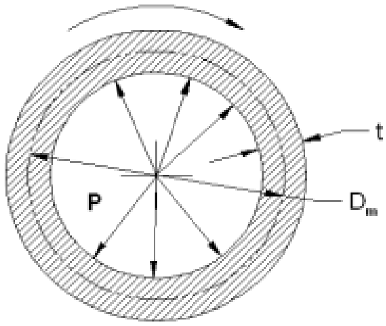

Fig.2 Hoop Stress in a Pressure Vessel

At the nozzle, the geometry changes, and the hoop stress distribution becomes more complex due to the stress concentration around the intersection [9].

Conventional Methods for Stress Analysis

This section outlines the methods for stress Analysis in pressure vessel nozzles. The methodologies used for this work are comparison stress analysis using three different methods of analysis:

Finite Elements Methods

The process of creating a model, which is a representation of how a system of interest is constructed and operates, is called modelling. A model is a simplified version of the system it depicts. A model's ability to help analysts forecast the impact of system modifications is one of its goals. A model should, on the one hand, closely resemble the real system and include the majority of its unseen characteristics.

However, it shouldn't be so complicated that it is hard to comprehend and use. Realism and simplicity are wisely traded off in a good model. The geometrical model of the Pressure vessel is shown in Figure 3.

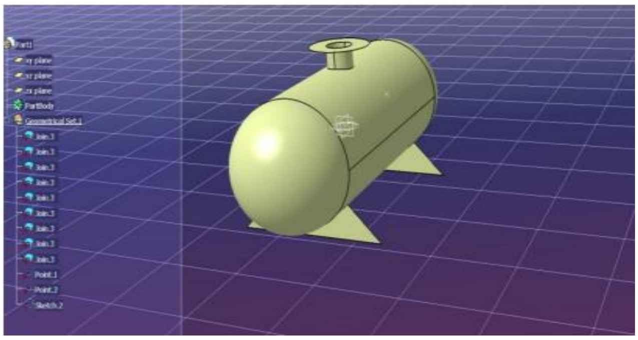

Fig.3 Geometrical Model of Pressure Vessel

Pre-Processing: Hyper mesh 9.0 is utilized for finite element modelling. Hyper Mesh is a powerful pre- and post-processor for finite element solvers that aids in the design process in a visually immersive and interactive setting. An essential part of finite element analysis (see in Figure 4) is choosing the right element type, order, and size.

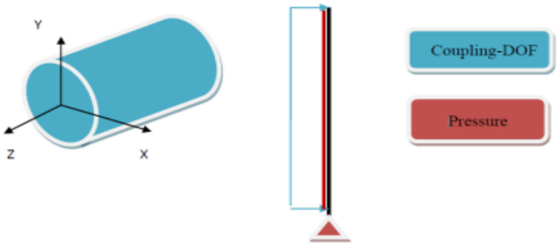

Fig.4 Axisymmetric FE Model

Element Type: Axisymmetric solid, shell/plate, or three-dimensional brick components are the ones used to build pressure vessels. However, compared to the other two, 3D components require a lot more processing time. The axis-symmetry of the vessel cannot be maintained while modeling it with the nozzle. Curved surfaces can be represented with shell components. Shell components are utilized since the vessel's thickness to radius (r) ratio is less than 0.1. Consideration of in-plane and normal loads, as well as moments and forces applied in all three directions by the nozzle, must be made while choosing the element type. Because of its bending and membrane properties, the Shell63 element finds widespread use.
Element order: Due to the relatively simple geometry, first-order components are utilized. The amount of time needed to calculate it is also lower than that of higher-order items [10].
Element size: The stress concentration will be at the junction of the nozzle neck and cylinder; hence it is represented with a denser mesh there. It examine the nozzle area curvature using chord deviation with varying element sizes. Initiating at the nozzle area, a 50-element size is chosen. Figure 5 displays the mesh model of the pressure vessel.

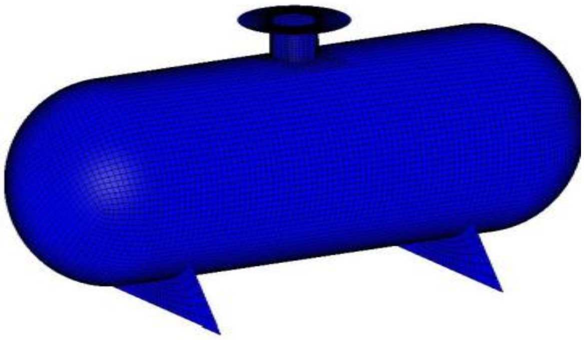

Fig.5 Finite Element Mesh Model

WRC 107 and, WRC 297: The WRC approach should not be applied outside of its scope. WRC 107's drawback is that stresses are only computed in the shell rather of the nozzle neck, even though it can be crucial for relatively small neck thicknesses. Shells with pressurized nozzles can employ WRC 297. Additionally, the WRC 297 approach may be used to calculate the stresses that external loads on nozzles cause. Keep in mind that the WRC 297 approach is typically seen as being unduly cautious when it comes to determining the stresses in the nozzle wall. The effects of any potential pressure stress intensification factor (PSIF) at an opening are not taken into account by the WRC 107 / 297 methodologies. In particular, when using WRC 107 and WRC 297 for nozzles, the stresses resulting from pressure must be computed in the shell or head [11].

Popular FEM Software for Pressure Vessel Analysis

ANSYS: ANSYS is a popular software used for finite element analysis (FEA), which is great for studying how pressure vessels handle stress. It's known for being flexible and powerful, making it a top choice for engineers. ANSYS is especially good at dealing with materials that don't behave in simple ways, such as those that stretch or deform over time, which is important for understanding how pressure vessels perform in the long run. ANSYS helps engineers see where stress is highest, find weak spots, and improve designs to make them safer and more efficient [12].
ABAQUS: ABAQUS is another widely used FEA software designed for the nonlinear and dynamic analysis and has traditionally been most effective for analyzing pressure vessels. Where stress concentration is highest, ABAQUS is especially effective in simulating contact stress. expansion due to heat and multi-axis load situations Common challenges in pressure vessels Stress analysis allows engineers to incorporate custom material behavior such as viscoelasticity or anisotropy for special applications [13].

Overview of Finite Element Methods (FEM)

Computational methods known as Finite Element Methods (FEM) are employed in engineering to resolve issues with complicated geometries, material characteristics, and boundary conditions. The approach divides a structure into discrete elements, each governed by mathematical equations, to approximate real-world behavior. Key concepts like meshing ensure the structure is broken into manageable parts, boundary conditions define constraints, and element types determine the accuracy of analysis. Historically, FEM evolved from solving structural problems in aerospace and mechanical engineering to being a versatile tool across disciplines.

Fundamentals of Finite Element Method (FEM)

Finite Element Analysis (FEA) is a numerical procedure which is employed to solve problems encountered in engineering sciences, whenever there is need to solve a partial or ordinary differential equation. It assumes the field variables as a discrete set at a certain point within a domain rather than continuous. FEM is Used to model geometries and boundary conditions which are not easily possible to solve by closed-form solution. Used in structural, thermal, fluid and electrostatic applications.

Basic Concepts of FEM

Domain: The continuous region under study (e.g., a beam or mechanical part).
Nodes and Elements: The domain is divided into small, simple shapes (elements), with nodes representing key points where field variables are computed.
Mesh: A combination of elements and nodes forming the basis for FEM.

Mathematical Foundation: The governing equations (e.g., $G(\phi) + f = 0$) are transformed into matrix equations, such as $[K]\{u\} = \{f\}$, where:

[K]: Property matrix (e.g., stiffness)
{u}: Behavior (e.g., displacement)
{f}: External forces or actions

Applications of FEM: Stress analysis, fluid flow, heat transfer, electrostatics, and dynamic analysis.

Key Concepts: Meshing, Boundary Conditions, an Element Types

There is some concept of finite element method to discuss below:

Meshing: The procedure for transforming non-standard forms into discrete volumes known as elements. To provide a reliable finite element analysis (FEA) simulation, meshing is an essential step.

Boundary conditions: A boundary condition can be Dirichlet, Neumann, Robin, mixed, or Cauchy. A appropriate solution is used to apply boundary conditions to the built equation.

Element Types: In finite element analysis, components are often classified as either 1D, 2D, or 3D. Some examples of element types include:

Continuum or solid element: The essential building block for meshes. The shapes can be either two-dimensional (quads and tris) or three-dimensional (hexahedral and tetrahedral).
Membrane element: Applied to surfaces that resemble thin fabrics [14].
Truss element: The element's axis is the loading point, and it is used to model structures that resemble lines[15].

Trends and Challenges in Finite Element Analysis: Nozzles and pressure vessels can't be adequately assessed without finite element analysis (FEA). Here are some trends and challenges associated with FEA in pressure vessel nozzles:
Cost: The cost of FEA has been decreasing due to improved software and lower hardware costs.
Modeling techniques: New techniques are being developed to automate meshing.
CAD systems: Many CAD systems can mesh finite elements, allowing engineers to quickly develop a model without re-entering geometry [16].
Design by analysis: This approach is more optimized than design by rule.
Hydrotest Testing: FEA can be used to check hydrotest results before the actual test.
Computational demands: FEA can be computationally demanding.
Mesh generation: Mesh generation can be intricate.
Software: FEA software needs to be advanced to handle more complex loading scenarios and real-time simulations.
Materials: Exploring FEA with new materials could lead to more innovative designs [17].
Stress distribution: FEA can help identify stress points and potential failure locations.
Boundary conditions: Care must be taken when applying boundary conditions [6].

Advantages of FEM in Structural Analysis

- The Finite Element Method has several benefits over other numerical approaches, including:
- Any form, load, and support conditions may be applied to the analyzed body.
- It is a very versatile method.
- The matrix mesh can mix elements of different types.
- Its versatility can be contained in a single program.
- The real building and its corresponding finite element model are quite similar [18].

Emerging Computational Techniques

The concept of AI-Assisted FEM brings presuppositions of new computational methods to translate the analysis of modelling in the engineering field. The incorporation of AI into FEM mostly follows techniques such as ANN and EPR this provide data-driven techniques to establishing constitutive modeling of materials.

Evolutionary Polynomial Regression (EPR): EPR employs evolutionary algorithms to construct polynomial representations of complex material responses, exclusively from experimental or field data without the requirement of a priori defined mathematical models.

EPR-based constitutive models are transparent, mathematically explicit and can be easily implemented into FEM software such as ABAQUS.

Artificial Neural Networks (ANN): ANNs are used to constitute model nonlinear material with the advantage of being able to predict very accurate stress-strain curve.

ANN models are capable of capturing the non-linear behavior of soils, rocks, and composites under different loading conditions with good accuracy and minimal computational effort.

However, ANN models require extensive training and optimal design of network architecture [19], which can be computationally intensive.

Applications:

- AI-Integrated FEM is deployed in problems of boundary values, when FEM cannot capture the nonlinearity of the material or its unpredictable behavior.
- In the present work, FEM can be enhanced with EPR and ANN, which means the models are modified with higher accuracy, thus increasing the potency of the method in the real environment [20].

FEM Applications in Pressure Vessel Nozzle Stress Analysis

FEM is widely used for analyzing stress in pressure vessel nozzles under various configurations and conditions. 2D FEM models are efficient for symmetrical nozzles with simple loading, while 3D FEM models are essential for capturing detailed stress distributions in complex geometries and multi-nozzle systems. FEM accurately simulates stress concentrations, flexibility, and interactions in diverse configurations, such as radial, tangential, and inclined nozzles.

2D vs 3D FEM Models for Nozzle Stress Analysis

Stress analysis utilizing photoelasticity will also be challenging due to the varied forms. The optimal method is clearly FEM stress analysis. As a result, the finite element method has been used for the study. When it comes to commercial finite element analysis software, one of the most prominent names is FEM from Ansys.

2D Model of Vessel with Nozzle Positioning

Figure 6 below shows the vessel's 2D model with the nozzle location. Now, let's analyze nozzle N1.

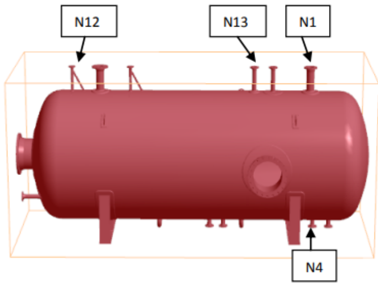

Fig.6 2D model of vessel with Nozzle positioning

Usage: Simplified models, often applied to axisymmetric nozzles under symmetrical loading conditions.
Advantages: Computationally efficient, faster to analyze, and suitable for preliminary studies.
Limitations: Unable to capture complex geometries, multi-directional loads, or detailed stress concentrations at intersections.
3D Model

The 3D model of carbon drain vessel is created in CAD and imported in Ansys software as shown in Figure 7 below.

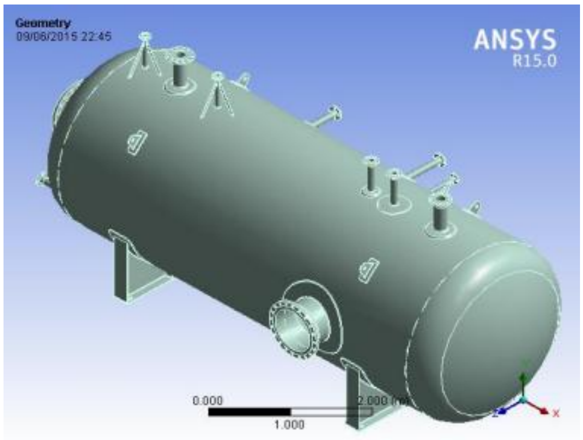

Fig.7 3D Model

Usage: Used for analyzing nozzles with non-axisymmetric geometries, multi-nozzle systems, or intricate loading conditions.
Advantages: Provides detailed stress distribution, captures geometric discontinuities, and handles combined thermal and mechanical loads.
Limitations: Requires more computational resources and time for meshing and solving [21].

Application of FEM in Various Nozzle Configurations

There are many kinds of application discuss below:

- Finite Element Method (FEM) is widely used for stress analysis and flexibility analysis in pressure vessel with different nozzle configuration.
- It can model accurately various nozzle types, such as radial nozzles, tangential nozzles or inclined nozzles and their orientation at cylinder shells or vessel heads.

- FEM can also consider the effect of geometrically or materially nonlinear key components under complex load conditions – internal pressure, thermal expansion, external forces due to connected piping systems etc.
- By using advanced meshing technique, FEM has the ability to locally refine meshes in critical region around nozzle intersections for accurate stress distribution determination.
- Also, FEM (Finite Element Method) is used to figure out bending stresses, check how flexible nozzle-to-shell connections are, estimate strength, and study how cracks grow.
- Special software like Nozzle-FEM and NozzlePRO helps in doing detailed checks on how safe and well nozzles perform in industries like oil and gas, petrochemical, and power generation.
- These tools help make nozzle designs better, increase safety, and make pressure vessels stronger in tough working conditions.

Analysis of Complex Geometries and Multi-Nozzl Systems

FEM is essential in modeling the stresses of complex geometries together with a multi-nozzle system. FEM models are used to provide stress patterns due to geometric discontinuities and interactions occurring at the nozzles. These high-stress concentrations result when these systems are subjected to combined internal pressure, thermal loading, and external forces from piping. Mostly, 3D FEM models are used to simulate such scenarios. Therefore, stress patterns can be revealed in detail, a situation not readily available when more nozzles are considered. Refinement of particular zones through destiny meshing techniques simulates these areas, which are later passed for further detailed analysis. Specific models defining meshes allow for the use of adaptive meshing for refining the critical regions at the applicability stage itself. This result has developed from the studies and lead to the setting of new instruction for the designs in order to be optimized for reducing their stress concentrations and thus increasing structural integrity of the pressure vessels [22].

Incorporating Material Nonlinearities and Thermal Loads

Material nonlinearity and temperature loads are severe factors in pressure vessel analysis due to their highly interacting effects on distributing the stress in the structures and their behavior un-der load. Thermal loads arising from temperature differences contribute predominantly to stress levels and at times it even exceeds that caused by mechanical loads alone.

Material Nonlinearities: Pressure vessels often operate under high pressures and elevated temperatures, where materials exhibit nonlinear behaviors such as plasticity and creep. Finite Element Analysis (FEA) effectively models these nonlinearities by incorporating material-specific properties like modulus of elasticity, yield strength, and thermal expansion coefficients.

Thermal Loads: Heat transfer through convection from fluids to the vessel wall results in thermal gradients, leading to thermal stresses. These stresses can be calculated using formulas like: thermal = $E \cdot \alpha \cdot \Delta T$, Where E is the modulus of elasticity, α is the thermal expansion coefficient, and ΔT is the temperature difference.

Geometric Effects: Vessel designs with uniform or optimized wall thickness reduce stress concentrations caused by thermal loads. Analysis showed that increasing wall thickness effectively reduces thermal and hoop stresses, improving overall structural stability.

Optimization and Results: Various pressure vessel configurations were studied, showing that thermal stress is a dominant factor. FEA results demonstrate that optimized vessel designs with reduced geometric discontinuities lead to significantly lower stress levels and improved safety [23].

Literature of Review

This section provides some previous research on Role of Finite Element Methods (FEM) in Pressure Vessel Nozzle Stress Analysis: A Survey of Applications and Trends of reviews are summaries in Table I, based on findings and limitations.

Bella, Bailey and Lu (2018) investigates the electrical and thermal properties of a circuit by means of the Finite Volume (FV) and Finite Element (FEM) approaches. Comparing the two approaches yields similar findings with respect to thermal gradients and joule heating. A decrease in size and an increase in power are necessary due to the growing number of thermal effects and the amount of heat produced by each device. There has been a steady uptick in the recent trend towards miniaturization and the integration of several functional blocks into a single, enormous system-in package or system-in chip [24].

Jie (2021) many mud pumps in land oil fields are mechanical pumps, and the pump speed is difficult to control. The finite element model of the casing head has established using finite element software, and the contact mechanics between the slip and the bush after the tail pipe hanger has seated, which has studied. The results show that the final hanging load of the hanger depends on the three factors, which include the casing shell itself, the ultimate weight of the internal hanger and the strength of the casing itself [25].

Bozkurt, Nash and Uzzaman (2020) the cylindrical vessel shells' nozzles are modelled as three-dimensional finite element systems, subjected to unit forces and moments. Some types of nozzles, such as the Steam Generator (SG), are fastened to pressure vessels in plants that use pressurized water for reactors. Nozzles with a non-axis-symmetric form are exemplified by the down comer feedwater nozzle in the top vessel shell and the economizer feedwater nozzle on the bottom vessel shell of the SG. Nozzles like this typically experience external stresses during plant operation, which include forces and moments [26].

Devarakonda (2023) requires accurate stress measurement to ensure pressure vessel structural integrity. To examine the distribution of stresses at the nozzle-vessel junction, FEA models and simulation methods are used. To ensure the accuracy of the model, the results from FEA are contrasted with theoretical computations. While expanding their understanding of stress behavior in pressure vessel nozzles, this study's findings shed light on how to enhance design while ensuring safe operation [17].

Fan and Hu (2023) analyzing local stress in the pressure vessel nozzle is often cumbersome and subjective, significantly affecting the speed and accuracy of analysis. To address this issue, this study has developed a plugin for calculating local stress in pressure vessel nozzle areas and validated its practicality through case studies. This research outcome provides a convenient and efficient tool for pressure vessel design and structural optimization, with the potential to reduce the complexity and time costs of the analysis process [27].

Niu et al. (2023) makes a significant contribution to FEM applications. According to the suggested study, equilateral triangular meshes exhibit the most consistent distribution of dispersion when subjected to the same order. The numerical findings demonstrate that with the same number of nodes and interpolation order. When using FEM, the findings are helpful in selecting the proper mesh form, node density, and element order [28].

Table 1 Presents the Comparative Table based on Big Data Analytics Using Cloud Computing

Reference	Study On	Approach	Key Findings	Challenges	Limitations
Bella, Bailey and Lu (2018)[24]	Electro-thermal behavior of an electronic device	Comparative study of Finite Volume (FV) and Finite Element Method (FEM)	Comparable thermal gradients and Joule heating results for both methods; highlights the importance of thermal effects due to device miniaturization and integration.	Increasing thermal effects with device miniaturization.	Limited to specific electronic device scenarios and lacks focus on nozzle stress analysis.
Jie (2021)[25]	Stress analysis in casing heads of mud pumps	Finite Element Modeling of casing heads to study contact mechanics	Identifies key factors influencing the final hanging load of the hanger: casing shell strength, hanger weight, and ultimate casing strength.	Applicability limited to mud pumps and mechanical stress analysis scenarios.	Does not address nozzle-specific stress analysis for pressure vessels.
Bozkurt, Nash and Uzzaman (2020)[26]	Nozzle stress analysis in cylindrical vessel shells	3D Finite Element Models of nozzles with applied unit forces and moments	Demonstrates stress distribution on steam generator (SG) nozzles, emphasizing external load effects during plant operations.	Non-axisymmetric shapes of nozzles complicate the analysis.	Focused on SG nozzles; does not generalize to other pressure vessel types or conditions.
Devarakonda (2023)[17]	Stress distribution in nozzle-vessel junction	FEA modeling and simulation techniques to analyze and validate stress behavior	Validates FEA results with theoretical calculations; provides insights into improving design and ensuring safe operations.	Precision in measurements can be difficult in real-world applications.	Limited exploration of external operational forces and conditions.
Fan and Hu (2023)[27]	Local stress analysis in pressure vessel nozzles	Development of a plugin for local stress calculation in pressure vessel nozzle areas	Provides a convenient, efficient tool for stress calculation, reducing analysis complexity and time costs.	Subjectivity in local stress interpretation before plugin implementation.	Plugin's applicability may vary across different pressure vessel designs and conditions.
Niu et al. (2023)[28]	FEM application and element properties in stress analysis	Study of equilateral triangular meshes for uniform dispersion in FEM analysis	Equilateral triangular meshes provide uniform dispersion under the same interpolation order; highlights importance of mesh choice in FEM applications.	Effectiveness depends on mesh quality and interpolation order.	Limited focus on nozzle stress analysis; primarily evaluates mesh properties.

Conclusion and future work

Finite Element Methods (FEM) are indispensable in analyzing pressure vessel nozzles due to their ability to handle complex geometries, multi-directional loads, and nonlinear material behaviors. 2D models are suitable for preliminary studies, while 3D models offer detailed insights into stress distributions and interactions in complex configurations. The incorporation of thermal loads and material nonlinearities further extends FEM's utility in simulating real-world conditions. FEM-based analysis has proven effective in optimizing nozzle designs, reducing stress concentrations, and ensuring compliance with safety standards. Despite its wide adoption, challenges like computational demand and modeling complexities remain areas for improvement.

Future advancements in FEM for pressure vessel nozzles should focus on integrating AI-driven techniques, such as machine learning models, to predict stress patterns and material behavior more efficiently. Enhanced meshing algorithms and adaptive techniques could significantly reduce computational requirements while improving accuracy in high-stress regions. Additionally, exploring hybrid approaches that combine FEM with experimental validations and incorporating emerging materials into simulations could further enhance the reliability of nozzle stress analysis. Collaboration across industries and academia will be crucial in developing standardized practices and software tools to address evolving engineering challenges.

References

[1] M. H. U. Sharif and M. A. Mohammed, "A literature review of financial losses statistics for cyber security and future trend," World J. Adv. Res. Rev., 2022, doi: 10.30574/wjarr.2022.15.1.0573.

[2] T. Fadiji, C. J. Coetzee, T. M. Berry, A. Ambaw, and U. L. Opara, "The efficacy of finite element analysis (FEA) as a design tool for food packaging: A review," Biosystems Engineering. 2018. doi: 10.1016/j.biosystemseng.2018.06.015.

[3] A. Cipollina et al., "Finite Element Analysis (FEA) of a Premaxillary Device: A New Type of Subperiosteal Implant to Treat Severe Atrophy of the Maxilla," Biomimetics, 2023, doi: 10.3390/biomimetics8040336.

[4] V. Thakaran, "A Comparative Study of Piping Stress Analysis Methods with Different Tools, Techniques, and Best Practices," Int. J. Adv. Res. Sci. Commun. Technol., vol. 2, no. 1, pp. 675–684, Oct. 2022, doi: 10.48175/IJARSCT-7868D.

[5] G. S. Kumar, "Role of Computational Tools in Structural Analysis and Design," vol. 24, no. 01, pp. 10090–10097, 2020, doi: 10.53555/V24I1/400285.

[6] H. A. Khande, A. B. Rathod, A. A. Khot, A. Afroz, and R. R. Mahajan, "Design and Finite Element Analysis for Pressure Vessel Nozzles," Jetir, vol. 6, no. 4, pp. 169–172, 2019.

[7] V. Thakran, "Environmental Sustainability in Piping Systems : Exploring the Impact of Material Selection and Design Optimisation," Int. J. Curr. Eng. Technol., vol. 11, no. 5, pp. 523–528, 2021.

[8] P. Narale and P. S. Kachare, "'Structural Analysis of Nozzle Attachment on Pressure Vessel Design,'" Int. J. Eng. Res. Appl., vol. 2, no. August, pp. 1353–1358, 2012.

[9] D. . Nnadi, C. . Eze, J. . Aririguzo, and C. . Agu, "Effect of Stresses on the Walls of a Pressure Vessel," Int. J. Eng. Trends Technol., vol. 48, no. 8, pp. 426–433, Jun. 2017, doi: 10.14445/22315381/IJETT-V48P274.

[10] V. V. Saidpatil and V. K. Kulloli, "Static, Linear and Finite Element Analysis of Pressure Vessel," IJLTEMAS, vol. III, no. IV, pp. 84–91, 2014.

[11] W. Stikvoort, "Pressure Vessel Design for Imposed Nozzle Loads," Int. Res. J. Mod. Eng. Technol. Sci., no. 10, pp. 821–823, Oct. 2022, doi: 10.56726/IRJMETS30669.

[12] R. Bahadur, V. K. Mittal, and S. Angra, "Stress Analysis of Pressure Vessel Nozzle using FEA," vol. 6, no. 16, pp. 1–6, 2018.

[13] H. Fan and L. Hu, "Pressure vessel nozzle local stress prediction software based on ABAQUS- machine learning," SoftwareX, vol. 24, Dec. 2023, doi: 10.1016/j.softx.2023.101550.

[14] H. S. Chandu, "A Survey of Semiconductor Wafer Fabrication Technologies: Advances and Future Trends," Int. J. Res. Anal. Rev., vol. 10, no. 04, pp. 344–349, 2023.

[15] D. Madier, "An Introduction to the Fundamentals of Mesh Generation in Finite Element," FEA Academy, 2023.

[16] K. Khadke and D. D. Chawde, "Design & Finite Element Analysis of Pressure Vessel," Int. J. Res. Appl. Sci. Eng. Technol., 2022, doi: 10.22214/ijraset.2022.46076.

[17] V. R. G. Devarakonda, "Finite Element Analysis (FEA) for Stress Evaluation Of Pressure Vessel Nozzles," Int. J. Core Eng. Manag., vol. 7, no. 5, 2023.

[18] I. D. Erhunmwun and U. B. Ikponmwosa, "Review on finite element method," J. Appl. Sci. Environ. Manag., vol. 21, no. 5, p. 999, 2017, doi: 10.4314/jasem.v21i5.30.

[19] H. S. Chandu, "A Survey of Memory Controller Architectures: Design Trends and Perforamance Trade-offs," Int. J. Res. Anal. Rev., vol. 9, no. 4, pp. 930–935, 2022.

[20] A. Javadi, M. Mehravar, A. Faramarzi, and A. Ahangar-Asr, "An Artificial Intelligence Based Finite Element Method," ISAST Trans. Comput. Intell. Syst., vol. 1, pp. 1–7, 2009.

[21] N. V Avhad and V. G. Bhamre, "Investigating Stress Level through FEA of Nozzles of Carbon Drain Vessel by ASME Sect-VIII," Int. Res. J. Eng. Technol., vol. 2, no. 5, pp. 918–921, 2015.

[22] N. K. Gautam, R. J. Patil, and S. S. Patil, "Investigation of Nozzle Shape, Number of Nozzles and Nozzle Inclination Angle and Its Optimization," Tuijin Jishu/Journal Propuls. Technol., vol. 44, no. 4, pp. 1266–1279, Oct. 2023, doi: 10.52783/tjjpt.v44.i4.1008.

[23] M. Ahmed, R. U. Khan, S. Badshah, and S. Jan, "Finite element investigation of geometry effect on pressure vessel under combined structural and thermal loads," Int. J. Eng. Adv. T echnology, vol. 4, no. 2, 2014.

[24] M. A. Bella, C. Bailey, and H. Lu, "Electro-thermal behaviour using finite volume and Finite Element method," in 2018 19th International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Microelectronics and Microsystems, EuroSimE 2018, 2018. doi: 10.1109/EuroSimE.2018.8369916.

[25] C. Jie, "Suspension Control Strength Analysis and Load Calculation Based on Finite Element Analysis," in 2021 13th International Conference on Measuring Technology and Mechatronics Automation (ICMTMA), IEEE, Jan. 2021, pp. 455–458. doi: 10.1109/ICMTMA52658.2021.00104.

[26] M. Bozkurt, D. Nash, and A. Uzzaman, "Effect of the Internal Pressure and External Loads on Nozzles in Cylindrical Vessel," IOP Conf. Ser. Mater. Sci. Eng., vol. 938, no. 1, p. 012007, Oct. 2020, doi: 10.1088/1757-899X/938/1/012007.

[27] H. Fan and L. Hu, "Automated Calculation Plugin for Local Stress in Pressure Vessel Nozzles," in 2023 2nd International Conference on Mechanical Engineering and Power Engineering (MEPE), IEEE, Dec. 2023, pp. 1–8. doi: 10.1109/MEPE60002.2023.10563183.

[28] Y. Niu, J. Liu, W. Luo, Z. Li, and J. Song, "Dispersion Characteristics and Applications of Higher Order Isosceles Triangular Meshes in the Finite Element Method," IEEE Open J. Antennas Propag., vol. 4, pp. 1171–1175, 2023, doi: 10.1109/OJAP.2023.3331217.