International Journal of Core Engineering & Management
Volume-7, Issue-05, 2023 ISSN No: 2348-9510

FINITE ELEMENT ANALYSIS (FEA) FOR STRESS EVALUATION OF PRESSURE VESSEL NOZZLES

Venkata Rama Gautham Devarakonda
Oxnard, CA, USA
Gautham.rama90@gmail.com

Abstract

Pressure vessels are critical components in various industries, including power generation, oil and gas, where they contain fluids or gases under pressure. Nozzles, essential for the inflow and outflow of fluids, introduce structural discontinuities in the vessel wall, necessitating rigorous stress analysis to ensure safety and integrity. The stress analysis of nozzles used in pressure vessels is examined in this work via the use of finite element analysis (FEA). Many different types of businesses rely on pressure vessels, and nozzles are an essential part of how these vessels work. However, the introduction of nozzles creates stress concentrations and discontinuities in the vessel wall. The work highlights the need of precise stress measurement in guaranteeing the structural integrity of pressure vessels. FEA modelling and simulation techniques are employed to analyse the stress distribution in the nozzle-vessel junction. Results obtained from FEA are compared with theoretical calculations to validate the model. Insights for improving design and guaranteeing safe operation are provided by the results of this research, which add to our knowledge of stress behavior in pressure vessel nozzles.

Keywords: pressure vessel, nozzle, stress analysis, finite element analysis (FEA), stress concentration.

I. INTRODUCTION

As an integral part of the infrastructure in many different sectors, pressure vessels have long been used for power production, oil and gas, and more. Heavy-duty accessories called pressure nozzles make it easier for these fluids to enter and exit the pressure vessel for further processing. It is necessary to make an aperture in the vessel and attach the nozzle to it in order to add the nozzle to it[1]. This process structurally breaks the vessel's shell, resulting in discontinuities in the wall of the vessel. Manufacturers need to make sure that any mistakes made during the nozzle weld do not compromise the pressure vessel's strength. Ultrasonic scanning combined with NDT is a reliable inspection technique to assess the condition of the welds supporting the pressure vessel nozzle. Nozzles for pressure vessels are an essential feature of every pressure vessel's design and analysis, especially so when FEA is used to determine the vessel's structural integrity.

An idealized physical issue may have its mathematical model roughly solved using a numerical approach called the finite element method (FEM) [2]. Discretizing the object into finite parts allows one to solve the resulting differential equations in either structural mechanics (algebraic equations) or dynamics (ordinary differential equations). Though FEM was initially developed to solve problems of structural mechanics it rapidly spread to other branches and areas such as heat transfer and fluid mechanics.

Gases and liquids may be stored at pressures greater than atmospheric pressure in sealed containers called pressure vessels. Parts of a pressure vessel include saddles, ladders, heads, base plates, flanges, nozzles, and internals. Stresses on the inside of a pressure vessel whose walls are relatively thin increase as the product of the vessel's radius and pressure grows. It is stated that an item has thin walls if its radius is more than 10 times its wall thickness. Pressure vessels rupture when their wall stress states surpass certain thresholds, highlighting the need to know and measuring stresses in these vessels [3]. The shear and normal stresses occurring on the walls of thin-walled pressure vessels are assessed in stress analyses of these vessels. A uniform distribution of stress throughout the thickness of the walls is one of the assumptions used when studying pressure vessels with thin walls, and here is where the concept of elasticity comes into play. The analysis holds water when the thickness is much less than the vessel's radius [4].

The FEA process for nozzle stress evaluation typically involves creating a detailed model of the nozzle and adjacent sections of the vessel, applying boundary conditions that mimic real-world operating environments, and then running simulations to assess the stress response. Engineers can evaluate the effects of different loading scenarios, such as internal pressure, external forces from connected piping, and thermal gradients [5]. The results of the FEA help in identifying stress concentrations that might not be apparent from simple calculations and allow for optimization of the nozzle design to enhance safety and performance. Additionally, FEA can be used to validate the design against code requirements, reduce the need for costly physical prototypes, and explore impact of different materials or design modifications on nozzle's stress behavior.

1.1 Research motivation and contribution

One goal of this study is to find the most important places for stress concentration and then use FER to improve the stress distribution in nozzles used in pressure vessels.. The goal of this effort is to identify future failure spots with enough accuracy to improve pressure vessel safety and dependability while lowering the requirement for expensive physical testing and guaranteeing industry compliance.

- This study provides an in-depth review of FEA techniques specifically applied to stress evaluation in pressure vessel nozzles, synthesising recent developments and methodologies in this area.
- The work compares traditional stress analysis techniques, such as WRC Bulletin methods, with advanced FEA approaches, highlighting advantages of FEA in terms of accuracy and computational efficiency.
- Detailed analysis of different meshing techniques and element types used in FEA for pressure vessel nozzles is presented, offering guidelines on selecting appropriate meshing methods based on the complexity of the geometry and required accuracy.
- Through case studies, the study validates FEA models against traditional hand calculations and other analytical methods, demonstrating the reliability of FEA in predicting stress behaviour in pressure vessels.
- The work identifies current challenges in applying FEA to pressure vessel nozzles, such as computational costs and meshing difficulties, and suggests future trends, including integration of ML techniques for enhanced analysis.

1.2 Structure of the paper

The arrangement of the paper is as follows: The introduction is covered in Section I, and a summary of FEA is given in Section II. The finite element approach is then shown in Section III. Nozzle design for pressure vessels is covered in Section IV. Next, Section V details the pressure vessel nozzle's stress analysis, and Section VI details the FEA model's work on the same. Lastly, sections VII and VIII provide the literature review conclusion and future work.

II. OVERVIEW OF FINITE ELEMENT ANALYSIS

Scientists, engineers, and mathematicians are the main users of FEA, a computer simulation method. It's a numerical method for forecasting a product's behaviour in response to actual physical factors such as vibration, heat, forces, and fluid flow. Failure, malfunction, or normal wear and tear of a product may be revealed using FEA. The precise creation date of FEA remains a mystery, although the idea behind it may be found in the seminal works of civil and aeronautical engineers Hrennikoff (1941) and Courant (1943), which were faced with the challenge of solving intricate structural analysis and elasticity issues. Computing power has greatly increased in recent years, allowing FEA to be used more often in large-scale projects [6].

2.1 Basic Principles of FEA

The main objective of engineering calculations in FEA is to gather data about how physical systems react to certain imposed circumstances in order to support and validate engineering choices and designs. The FE approach has been around for a long time, yet it's still the go-to technique for structural assessments. Not only is FEA becoming increasingly popular and widely utilised in the field of structural mechanics, but it has also found success in other domains including fluid dynamics, heat transport and conduction, electric and magnetic fields, food processing, and packaging, to name a few [6].

2.2 FEA Software and Tools

A physical item is divided into many thousands or even millions of finite elements, which are represented by shapes like cubes or tetrahedrons, using FEA software. Calculus formulas aid in predicting each element's behaviour. To forecast the behaviour of the real item, a computer then averages or sums up each distinct behaviour.

FEA software closes the gap between your 3D design and the real world. When you can apply forces like heat, mechanical stress, and vibration computationally to your digital model, its performance in the real world can be modelled with a deep level of accuracy.

FEA software lets you prototype virtually, addressing each area of lower-than-acceptable performance in turn (or all at once) to improve your model before you ever commit time and resources to make a physical prototype [7].

2.3 Meshing Techniques and Element Types

Mesh is a crucial step in executing a solid FEA model. Nodes, which may be anywhere in space depending on the kind of element, define the geometry of a mesh and its constituent parts, the elements. Since FEA solvers struggle with non-standard shapes, the meshing approach entails converting these forms into more recognizable volumes called "elements."
Two primary approaches are used for meshing. Here, we're talking about 3D models:

1. Tetrahedral element meshing or “Tet”
2. Hexahedral element meshing or “hex”

The findings are often more accurate when using hex or "brick" elements at lower element counts compared to Tet elements. If the geometry is really complex, Tet elements could be something to explore.

1. Hybrid meshing

This method integrates hex and Tet components in a manner that allows you to mesh different parts of the geometry using different techniques. This might lead to a decrease in the amount of geometry preparation required while increasing the number of local control meshes.

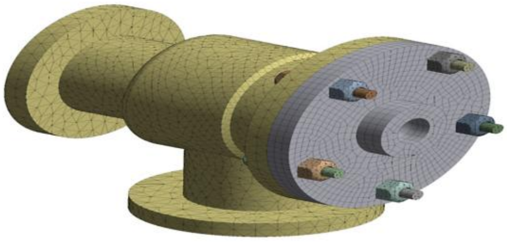

Figure.1: A pipe connection geometry using hybrid meshing

2. Sweep Meshing

Sweep meshing is a technique that helps build efficient meshes with regular sizing by physically moving the mesh across the volume and faces.

The amount of precision you aim for, the kind of analysis (explicit or implicit), and the physics you're solving for are standard factors to consider when choosing a mesh approach. Additional choices include layered tets and cartesian meshing, which are used in targeted analyses such as additive manufacturing [8].

III. FINITE ELEMENT METHOD

There are several approaches that may be used to numerically approximate a continuous solution field u of any PDE provided by Eq. (1) on a certain domain Ω. These approaches are some of the most popular ones: particle methods, finite-element methods, finite-volume methods, and finite-cell methods. Limiting ourselves to Galerkin-based finite element techniques, we provide this contribution.

$$L(u)=0 \quad \text{on } \Omega \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (1)$$

$$u=u\_d \quad \text{on } [\![\, \Gamma \,]\!]\ \_d \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ (2)$$

$$\partial u/\partial x=g \quad \text{on } [\![\, \Gamma \,]\!]\ \_N \ldots\ldots\ldots\ldots\ldots\ (3)$$

Think about the partial differential equation (PDE) in (1), the boundary conditions provided by (2) and (3), and the domain Ω on which it is defined. Here, the corresponding boundary conditions on the Dirichlet and Neumann borders are represented by and g. Equation (4) represents the system of equations that result from discretizing a domain with m elements and n nodes, adding boundary conditions, and then using a finite element version of equation (1). We consider that every requirement on the test and trial spaces is met.

$$\underbrace{\begin{pmatrix} k_{1,1} & k_{1,2} & \cdots & k_{1,n} \\ k_{2,1} & k_{2,2} & \cdots & k_{2,n} \\ \vdots & \vdots & \ddots & \vdots \\ k_{n,1} & k_{n,2} & \cdots & k_{n,n} \end{pmatrix}}_{K(u^h)} \underbrace{\begin{pmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{pmatrix}}_{u^h} = \underbrace{\begin{pmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{pmatrix}}_{F} \ldots\ldots\ldots\ldots (4)$$

As the stiffness matrix, a non-linear left-hand side matrix in Eq. (4) is denoted as K(uh). where uh denotes a discrete solution field and F stands for the right-hand side vector. Equation (4)'s system of equations residual may be expressed as

$$r(u^h) = K(u^h)u^h - F \ldots\ldots\ldots\ldots\ldots \text{(5)}$$

The solution u^h, may be obtained by using a Newton-Raphson iteration approach with the tangent matrix and the linearization of r(u^h). In each iteration, a linear system of equations must be solved. Continue repeating this process until the residual norm rn satisfies the tolerance criteria. See here for an analysis of the methods in depth. It takes exactly one iteration for this formulation based on residuals to converge when dealing with a linear operator L. Of all the procedures in finite element methods, solving the linear system of equations is the most computationally intensive for a large number of elements and nodes. It is very necessary to skip this phase when dealing with applications that rely on efficient processing, such as digital twins and real-time simulations. To drastically cut this cost, methods that are well-suited to these kinds of applications, such as model order reduction, build a surrogate model of Eq. (4). This cost can be completely avoided with neural network methods; however, a lot of training and test data is needed, which is usually made by modelling the finite element problem that the neural network is modelling. In this case a residual r(u^h) becomes

$$r(u^h) = Ku^h - F \ldots\ldots\ldots\ldots \text{(6)}$$

Section "Finite element method-enhanced neural network for forward problems" details a method for training a neural network to solve linear PDEs by combining numerical method residual information with additional training data [9].

IV. PRESSURE VESSEL NOZZLES: DESIGN

A pressure vessel's nozzle is the aperture via which fluid may be introduced or removed. One common method of connecting pipes or pieces of machinery is to extend the nozzle from the pressure vessels outside. Such nozzle connections are essential for the pressure vessel to operate correctly.

The pressure retaining wall is penetrated after an aperture is cut in the pressure vessel to attach the nozzle to it. Consequently, it causes a break in wall of the pressure vessel by weakening the boundary. Both outside and inside of the pressure vessel may be perforated to accommodate nozzles. Three components make up a pressure vessel nozzle, which is also shown in figure 2.

- A flange Connection (for flanged connection with pipe)
- Nozzle Neck part and
- Reinforcement (in case required)

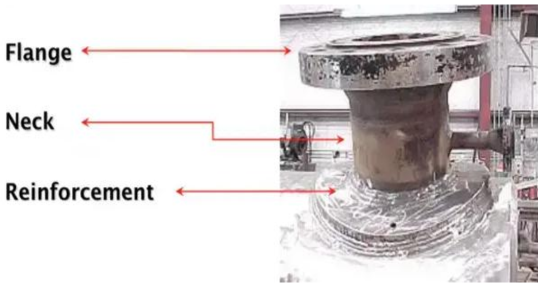

Figure. 2: Elements of a pressure vessel Nozzle

Pressure vessel nozzles may be broadly spitted into two categories [10]:
1) Radial Nozzle and
2) Non-Radial Nozzle

- Hill Side Nozzle and
- Tangential Nozzles
- Angular Nozzles
- Shell Nozzles
- Head Nozzles

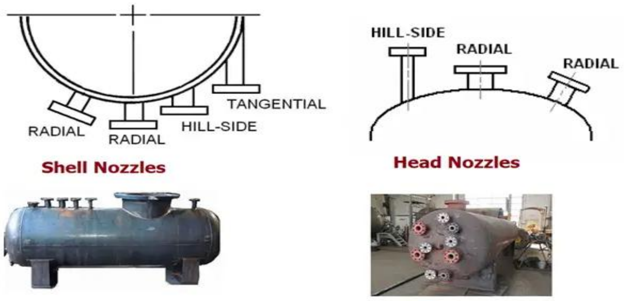

Fig. 3: Pressure Vessel Nozzle Types

Figure 3 shows the types of nozzles attached to pressure vessels in, which figure shows radial, hill-side and tangential nozzles.

V. STRESS ANALYSIS OF PRESSURE VESSEL NOZZLE

Each component of the pressure vessels is stressed to varying degrees by a number of loads. The kind of load, the geometry of the vessel, and the design of its components all influence the categories and intensities of stress. According to ASME VIII division 1 paragraph UG-22, pressure vessels are subject to loads caused by a variety of factors, including internal and external pressures, the weight of equipment, static reactions, welded components (including nozzles, pipes, isolations, and internal supports), temperature variations, wind loads, seismic forces, fluid impact reactions, and temperature gradients [11].

5.1 Stress Categories

Three levels of stress are distinguished: primary, secondary, and peak. Bending and membrane load are two broad categories into which the principal stresses fall. Membrane and bending stresses include the secondary stresses.

5.1.1 Primary stress

Shear stress produced by loading or normal stress might be the main source of stress. When the structure flexes, the levels of these stresses do not diminish. Primary bending stress (Pb), primary local membrane (PL), and primary general membrane (Pm) are the three main types of primary stress.

5.1.2 Secondary stress

When a structure undergoes deformation, the value of secondary stress, which might take the form of either normal or shear stress, decreases. There are two types of secondary stresses, denoted by the letter Q: secondary bending stress and secondary membrane stress. Thermal expansion-induced loads and moments are instances of Qm, secondary membrane stress. The process of connecting the body flange to the shell might lead to additional bending stress. Bending stress caused by loads and moments in supports and nozzles is another example.

5.1.3 Peak stress

A stress that is cumulative is the peak stress, denoted as F. At the place where stress is concentrated, the main and secondary stresses add up to peak stresses. Fatigue situations or brittle materials are the only ones where peak stresses matter. Fatigue cracks may be caused by peak stresses, which include membrane, bending, and shear stresses. Discontinuity corners are one example of a location where tension may build up.

5.2 Localized stress in nozzle

Nozzles subjected to external forces, including those exerted by pipes, cause bending stress, shear stress, and additive membrane stress in spherical and cylindrical shells, respectively. According to Bulletin WRC 107 and as shown in Figure 4, the sign system and nomenclature of local loads.

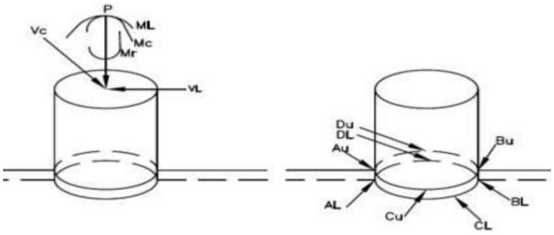

Figure 4: Convention and Nomenclature of Local Loads

5.2.1 **Evaluation of Stress Analysis**

Specifically, WRC Bulletin 107 and WRC Bulletin 297 outline the primary methods for assessing local loads. However, local loads are also evaluated using finite element techniques because of the restrictions imposed by these bulletins.

- Bulletin WRC 107- The parameterised process of stress calculation from the nozzle is outlined in Bulletin 107. A dimensionless set of input values yields stress outcomes due loads derived from experimental data-driven curves.
- Bulletin WRC 297- WRC Bulletin 297 provides information for wider diameter-thickness ratios than WRC 107, making it a complement to WRC 107.
- Finite Element Analysis- One approach to finding a numerical approximation for a given issue is the finite element technique. Nodes are common points in a network, and they include things like vertices, middle points, and the element's middle point. The domain geometry vividly represents the solution as a result of this decision. With its distinctive static materials and boundary conditions, this approach is well-suited for static analyses

involving known pressures or displacements and static loads. Analysis using the finite element technique requires certain data, including geometry, elements, materials, boundary conditions, loads, and solution methods. Among pressure vessels, 3D elements are the most prevalent. Theoretical foundation of elasticity for solid element [8]. For every 3D solid element are defined 3 degrees of freedom: orthogonal displacement: u_x, u_y, u_z [11].

VI. FEA MODEL FOR PRESSURE VESSEL NOZZLES

A finite element tool called ANSYS was used to model the internal pressure and external loading limit load analysis. An precise and dependable FEM for nozzle-cylinder interactions was shown and detailed in [12]. South American carbon steel plate SA-516 Grade 70 was the material to be examined for the pressure vessel's exterior. A youthful modulus of 200 GPa and a Poisson ratio of 0.29 characterize this material. To be more precise, the carbon steel used to make the nozzle was grade 2. For the material used to make the nozzle, the Poisson ratio and young modulus were both 0.29. There was a yield stress of 260 MPa for the shell and 290 MPa for the nozzle.

Furthermore, a comprehensive parametric investigation on FEM has been presented, along with a high-fidelity model methodology. Results from the model validation study were highly concordant with those from previous research (roughly 1% difference). There was also considerable agreement when comparing the parametric findings produced with the same model to WRC 537.

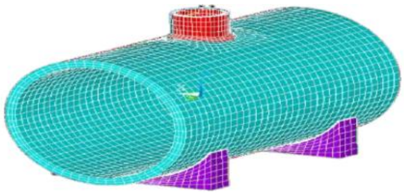

Figure 5: Perspective view of the meshed model

Figure 5 shows a cylindrical pressure vessel with a nozzle connected to its side in a 3D finite element model. Two supports hold the vessel steady. A mesh of varying colors covers the model, each colour possibly indicating a particular set of material attributes or analytical findings.

6.1 Verification of the Model

Analytical and finite element techniques' outputs were used throughout the model verification process. This outcome was achieved by a three-step process. The maximum allowable internal pressure of the model is first checked. After that, we checked the state of the maximum tensile loading. At last, the moment effect due to nozzle local loads is considered. Detailed explanations of the verifications follow. The shell's yield stress (σy), inner radius (r), and thickness (mm) were all measured at 250 mm and 33.5 mm, respectively, for all validation studies. In addition, the nozzle had a yield stress (σy) of 290MPa, a shell inner radius of 62.5mm, and a nozzle thickness of 15mm.

$$First\ yield:\ p_y = \frac{\sigma_y}{2}\left(1 - \left(\frac{r_i}{r_o}\right)^2\right) \ldots\ldots\ldots\ldots\ldots\ldots \ (7)$$

$$Limit\ Load:\ p_1 = \sigma_y \ln\left(\frac{r_i}{r_o}\right) \ldots\ldots\ldots\ldots\ldots\ldots \ (8)$$

There was only one nozzle cylindrical vessel that was exposed to internal pressure in this verification investigation. There was little mobility below the saddle due to the adoption of an open-ended model. We found that the saddle and nozzle had a friction force of 0.45. Using the Perfect Plasticity and Tresca criteria, a force and limit needed to achieve an initial yield may be calculated using equations (7) and (8). Since the shell's yield point is lower than the nozzle's, we want to use this information to find a limit load within a shell.

Table 1. Comparison the limit load results for internal pressure and external loading

Loading Type	Hand Calculations	FE Results	Difference
Internal Pressure	32.604MPa	30.333MPa	6.90%
Moment Effect	6773.17kN	6640kN	1.96%
Tensile Loading	19132kN	19333kN	1.04%

Table 1 shows that there is a discrepancy of around 2% between the numerical findings and those produced using the finite element model [13].

VII. LITERATURE REVIEW

Pressure vessel apertures analyzed using the limit-load approaches have been the subject of substantial study by a large number of researchers thanks to developments in finite element technology. In [14], The primary focus of structural integrity design criteria is to reduce the likelihood of collapse of offshore equipment when subjected to external loads. Cylindrical components undergo early plastic deformation in the form of buckling, and the last sites of collapse are often situated close to the connected portion when the cylinder and both ends are secured by welding or bolting. In other words, models of pressure vessels made of stainless steel were intended to withstand pressures up to 1000 meters while models made of aluminium alloy were intended to withstand pressures up to 2000 meters. The FEM code's eigenvalue analysis and empirical equations were used to carry out the pressure vessel design method. The findings were confirmed and studied by pressure testing conducted in KRISO's hyperbaric chamber.

In [15], cylindrical pressure vessels supported on two saddles and subjected to different boundary conditions may have their estimated stresses calculated using a finite element method. After determining stresses in a vessel, several load situations are taken into account, and static structural analysis is performed. By adjusting the component thicknesses of the vessel, a maximum von-misses stress is contained within a restriction. Within specific bounds, the maximum stress decreases as component thickness rises.

In [16], Workbench finite element approach developed the finite element model. The multi-layer composite cylinder's prestressed condition and operational state were computed using the finite element paradigm. A multi-layer ultrahigh-pressure cylinder with an optimized design has been created. The findings demonstrate that there is significant room for improvement in the cylinder's no uniformity of stress distribution.

This study [17], studies the impact of winding angle on COPVs made using the filament winding procedure, which involves winding a liner with continuous fibres impregnated with resin. The carbon T300/epoxy material is taken into account in the COPV investigation. The thickness of the composite vessel is determined via netting analysis. The research concentrated on failure analysis, total deformation, stress production, and ideal winding angle of composite pressure vessels. Tsai-Wu failure criteria are used to forecast COPV failure. The analytical technique takes into account the CLT and failure criteria; to validate, it compares the acquired findings to numerical data derived from the ANSYS workbench (ACP).

In [18], may be used to do a stress study on a particular steel high-pressure tank with two layers and a spherical shape. It all started with making the finite element model in ANSYS. An adaptive meshing approach produced the correct mesh density distribution, which was necessary to attain an acceptable analytical precision. The results provide a useful recommendation for its redesign.

This study [19], offers a straightforward approach to failure analysis using limit and shake-down methods for pressure vessels with flaws. The flaws that are being examined include part-through slots that have different geometric patterns. In this context, the engineering problem is relevant to the pressure vessel sector. The outcomes are contrasted with those that are produced by semi empirical formulas used in industry and, where feasible, with those that are derived by a methodical process employing the professional code ABAQUS. The results show that compared to marching solutions obtained by step-by-step elastic-plastic analysis, the limit and shakedown analysis approaches are more expensive and dependable. Pressure vessel load-carrying capacity are studied in relation to different part-through slots.

In [20], An aluminum-lined pressure tank composed of CFRP is modelled using the ANSYS finite element software. The outside filament wound fibres are wrapped using a combination of hoop winding and helical winding. Working at pressures more than 35 MPa makes safety a top priority. Each unidirectional layer in the vessel section has its own variation in thickness and wrap angle. A static inspection is performed on the vessel. It is possible to estimate the vessel's burst pressure using the maximum stress criterion. Table 1 provides a clear overview of the methodologies, results, gaps, and potential areas for future research in the field of pressure vessel analysis.

Table 1: Summary of related work for pressure vessel analysis based on FEA

Reference	Methodology	Results	Research Gaps	Future Work
[14]	Empirical formulae and eigenvalue analysis using FEM code; pressure tests in a hyperbaric chamber.	Empirical and FEA results were overestimated compared to pressure tests.	Need for improved accuracy in predictive models.	Refine empirical and FEM models to better match experimental results.
[15]	FEA on cylindrical pressure vessels with saddle supports under various load cases.	Increased thickness reduces stress within limits; stresses were calculated under different boundary conditions.	Limited exploration of dynamic or fatigue loading scenarios.	Extend analysis to include fatigue, dynamic loading, and real-time operational conditions.
[16]	FEA on multilayer composite cylinders using prestressed and working state conditions.	Non-uniform stress distribution; optimization improved stress uniformity.	Limited to static analysis; lack of experimental validation.	Include experimental validation and explore dynamic load conditions.
[17]	FEA on Composite Overwrapped Pressure Vessel (COPV) using classical laminate theory (CLT) and Tsai-Wu failure criteria.	Optimum winding angle identified; results validated against ANSYS Workbench.	Consideration of more complex failure mechanisms and environmental factors.	Expand the study to include environmental effects and multi-layer composite analysis.
[18]	FEA on two-layered spherical high-pressure vessels using adaptive meshing in ANSYS.	Identified weak zones and explosion pressure; provided redesign suggestions.	Need for real-world validation and consideration of multi-material effects.	Perform experimental tests and include multi-material layer analysis.

[19]	Limit and shakedown analysis for pressure vessels with part-through slots; compared with ABAQUS and semi-empirical formulae.	More economical and reliable methods were found; the effects of slots on load-carrying capacity were evaluated.	Limited to specific defect types; real-world operational conditions not considered.	Extend analysis to other defect types and validate with real-world data.
[20]	FEA on CFRP pressure vessels with aluminum liner, analyzing filament winding techniques.	Burst pressure is predicted using maximum stress criteria.	Lack of consideration for environmental and long-term durability factors.	Investigate environmental effects and long-term durability of CFRP pressure vessels.

VIII. CONCLUSION AND FUTURE DIRECTION

The significance of FEA in determining a stress distribution in pressure vessel nozzles has been highlighted in this study. This analysis is critical for guaranteeing the structural integrity and safety of pressure vessels in different sectors. FEA has proven to be superior to traditional stress analysis methods, especially in dealing with complex geometries and diverse loading conditions. The study also validates FEA's accuracy and reliability, reinforcing its significance in modern engineering applications. However, the application of FEA is not without challenges, particularly in terms of computational demands and the intricacies of mesh generation.

Future research in this area should aim to overcome the current limitations of FEA by developing more efficient and scalable algorithms that reduce computational costs. The integration of ML and AI could revolutionize predictive analysis, making it possible to anticipate stress behavior under varied and unpredictable conditions. There is also a need for advancing FEA software to enable real-time simulations and to handle more complex loading scenarios, including multi-material and multi-physics environments. Moreover, exploring the application of FEA in new and emerging materials could further enhance design and safety standards of pressure vessels, leading to more innovative and resilient engineering solutions.

REFERENCES

1. S. H. Jeong, K. S. Chung, W. J. Ma, J. S. Yang, J. B. Choi, and M. K. Kim, “Thermal stress intensity factor solutions for reactor pressure vessel nozzles,” Nucl. Eng. Technol., 2022, doi: 10.1016/j.net.2022.01.006.
2. C. H. Lee and H. W. Chou, “Stress intensity factor assessment for reactor pressure vessel nozzles containing postulated corner cracks,” Eng. Fract. Mech., 2022, doi: 10.1016/j.engfracmech.2022.108838.
3. M. Giglio, “Fatigue analysis of different types of pressure vessel nozzle,” Int. J. Press. Vessel. Pip., 2003, doi: 10.1016/S0308-0161(02)00151-5.
4. 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.
5. M. Alhijazi, Q. Zeeshan, Z. Qin, B. Safaei, and M. Asmael, “Finite Element Analysis of Natural Fibers Composites: A Review,” Nanotechnology Reviews. 2020. doi: 10.1515/ntrev-2020-0069.
6. 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.
7. “How does finite element analysis work?,” autodesk.com.
8. “The Fundamentals of FEA Meshing for Structural Analysis,” ansys.com.

9. R. E. Meethal et al., “Finite element method-enhanced neural network for forward and inverse problems,” Adv. Model. Simul. Eng. Sci., 2023, doi: 10.1186/s40323-023-00243-1.
10. “Pressure Vessel Nozzles: Definition, Types, Allowable Loads and Design,” whatispiping.com.
11. T. Lima, W. Andrade de, and P. Américo Almeida Magalhães, “Analysis of Stress in Nozzle/Shell of Cylindrical Pressure Vessel under Internal Pressure and External Loads in Nozzle,” J. Eng. Res. Appl. www.ijera.com ISSN, 2015.
12. M. Bozkurt, D. Nash, and A. Uzzaman, “Investigation of the stresses and interaction effects of nozzle-cylinder intersections when subject to multiple external loads,” in American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP, 2019. doi: 10.1115/PVP2019-93306.
13. M. Bozkurt, D. Nash, and A. Uzzaman, “Effect of the internal pressure and external loads on nozzles in cylindrical vessel,” in IOP Conference Series: Materials Science and Engineering, 2020. doi: 10.1088/1757-899X/938/1/012007.
14. J. M. Kim, H. J. Choi, S. G. Lee, T. H. Joung, and J. H. Lee, “Comparison of hydrostatic pressure test and finite element analysis results of cylindrical pressure vessel with various thicknesses,” in OCEANS 2019 - Marseille, OCEANS Marseille 2019, 2019. doi: 10.1109/OCEANSE.2019.8867263.
15. S. V. Dubal and S. Y. Gajjal, “Finite element analysis of reactor pressure vessel under different loading conditions,” in Proceedings - 1st International Conference on Computing, Communication, Control and Automation, ICCUBEA 2015, 2015. doi: 10.1109/ICCUBEA.2015.12.
16. W. Xinjie, L. Feiyi, Y. Jiansong, and W. Caidong, “Finite element analysis and optimization of multi-layer combined ultrahigh pressure cylinder,” in 2018 IEEE 9th International Conference on Mechanical and Intelligent Manufacturing Technologies, ICMIMT 2018, 2018. doi: 10.1109/ICMIMT.2018.8340432.
17. S. T. Atul, S. S. Bhat, S. S. Chavan, S. B. Kamble, A. P. Kulkarni, and S. B. Sangale, “Finite element analysis of composite overwrapped pressure vessel for hydrogen storage,” in 2016 International Conference on Advances in Computing, Communications and Informatics, ICACCI 2016, 2016. doi: 10.1109/ICACCI.2016.7732083.
18. S. Yu, Y. Yin, B. Xu, Y. Tan, and Y. Sun, “Finite element analysis for a two-layered spherical high-pressure vessel,” in ICCMS 2010 - 2010 International Conference on Computer Modeling and Simulation, 2010. doi: 10.1109/ICCMS.2010.145.
19. B. Yang, G. Chen, Z. Xu, and B. Xu, “Shakedown and limit analysis of defective pressure vessels,” Tsinghua Sci. Technol., vol. 6, no. 4, pp. 326–330, 2001.
20. Y. Wang, M. Sun, Z. Zheng, and S. Zhu, “Finite element modeling of carbon fiber reinforced polymer pressure vessel,” in ICENT 2010 - 2010 International Conference on Educational and Network Technology, 2010. doi: 10.1109/ICENT.2010.5532177.