International Journal of Interdisciplinary Innovative Research & Development (IJIIRD)
ISSN: 2456-236X
Vol. 10 Issue 01 | 2025

Design and Vibration Analysis of Multiphysics Stress on Pressure Vessel in Low Pressure and Medium Temperature Working Conditions

Mr. Shailendra R. Gawand[1], Prof. N. A. Kharche[2], Prof. S. R. Shekokar[3], Prof. P. V. Chopde[4]
[1]Researcher, Department of Mechanical Engineering, Padm.Dr.V.B.Kolte College of Engineering, Malkapur
[2,3,4]Assistant Professor, Padm.Dr.V.B.Kolte College of Engineering, Malkapur. Maharashtra India.

DOI: 10.5281/zenodo.16403151

ABSTRACT

The work focus on FE analysis and investigations of thermo mechanical constraints on the pressure vessel. The thermal stresses, deformation developed in the finite element analysis (FEA) for the pressure vessel are compared with analytical results. The vessel is designed using ASME section VIII Division 2 and the required thickness of the head, cylindrical shell etc. are to be validated using the FEA tool. The static and thermal analysis (multi-physics analysis, combined loading) will be carried out using ANSYS tool. Also natural frequency of pressure vessel will found out using modal analysis. Study focuses on two different constraints conditions of the right-hand saddle and performed for constant internal design temperature and internal design pressure.

Key words: Stress analysis, pressure vessel, vibration

1. INTRODUCTION

1.1 Introduction and background

A pressure container is a device designed as a closed container that keeps gases or liquids under pressure that differs considerably from ambient pressure. Due to the different operating conditions of vessels under pressure, they are potentially dangerous and accidents in which they can be deadly and pose fatal risks. The main purpose of this work is to design and analyse a pressurized tank to work under different operating conditions and to identify the most contributory parameter that regulates the efficient operation of the oil tank. In general, the pressure will develop in the oil tank, and will also withstand more forces deployed because of the internal and external pressure working on it, making the design critical. As a result, the pressure vessel is designed for safety reasons in accordance with ASME standards. Additional validation of the analysis software was created by comparing the results with the results of the manually calculated design values. We are working to understand the different loads that have developed in the ship under pressure. Creating a 3D model and analyzing using an appropriate solution. The result was compared with the results of the analysis software and turned out to be with a pleasant range. A Pressure Vessel is a container which contains either external or internal pressure, by the heat applied from direct, indirect or combination of sources. Design of pressure vessel is based on standard technical specification [ASME Code, Section VIII].

Pressure vessels are used for various purposes such as nuclear reactor vessels, pneumatic reservoirs and storage vessels of liquefied gases (Lee et al., 2017). From last few decades due to increased demand of alternative fuels generates the need of high pressure and temperature vessels for petroleum refineries and chemical plants. Pressure vessels are containers used to handle fluids which are highly toxic, compressible and works at high pressures. These vessels are applied in numerous industries such as oil, gas, petroleum, beverage, chemical, power generation, food and fertilizer, etc. Currently there is much advancement in the pressure vessel field like in case of investigating new grade material, composite materials, welding techniques, etc. The applications of finite element analysis is important for understanding of fatigue and creep process (Raffiee et al., 2018). In some chemical industry mixing of liquid or gaseous chemical take place in enclosed chamber such as pressure vessel hence it is called as mixing chamber. For deeper understanding of stress can be archived by multi-physics analysis. The multi-physics analysis is used to determine structure under the action of any combine loads. It is used to determine displacements, strains and force of a component. Hence in present investigations an attempt is made to carry out stress analysis to determine the stresses in structure under the action of static and thermal loads.

In pressure vessel during the operation chemicals comes through the nozzle with different pressure and temperature which varies with respect to time. The change in pressure and temperature results deformation and distortion with high local stresses in mixing chamber which in turns reduces the fatigue life of mixing chamber. The transient dynamic analysis is used to find stress and deformation in the pressure vessels. Hence in present investigations an attempt is made to carry out stress analysis to determine the stresses in structure under the action of any general loads. Also the fatigue life is predicted and enhanced by finite element analysis to determine the time varying displacements, strains and force acting on pressure vessels. The various types of pressure vessel are discussed in next section; the Figure 1 shows the generalized diagram of pressure vessel.

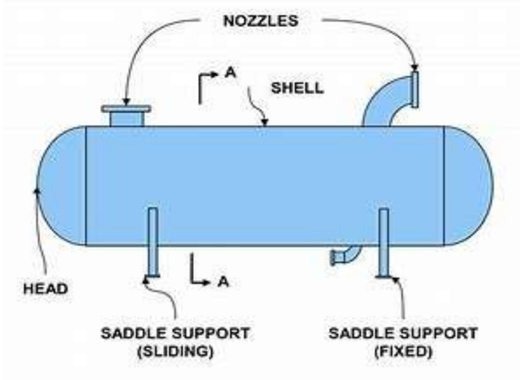


Figure 1: Different components of pressure vessel

1.2 Need

Understanding the design and manufacturing process of the pressure vessel is also crucial to keep the working environment safe. Improper design can lead to hefty fines and other legal complications. Due to internal pressure and temperature of chemical causes deformation in pressure vessel. Hence, it will be failed before the expected fatigue life. To design virtual vessel and analysis in ANSYS which save the manufacturing cost. Finite element simulation to minimize physical testing and possibility of failure in vessel is reduced.

2. LITERATURE SURVEY

As per the stated problem the work focused on this area is reviewed with the help of standard journal papers. The survey of literature based on the FEM as follows-

2.1 Literature survey

Apurva R.Pendhaje et al (2014) reported that following these standards leaves the designer free from designing the components. This aspect of design greatly reduces development time for a new pressure vessel.
Adithya.M and Patnaik.M. (2013) carried out FEM analysis and reported that design by analysis is most desirable to evaluate and predict the behavior of different configuration of PV supported on saddle with/without stiffener rings.
Vishal V. Saidpatil and Arun S. Thakare (2014) studied design and analysis of PV used in boiler for optimum thickness, temperature distribution and dynamic behavior using FEM. Farhad Nabhani et al (2012) conducted the work on Reduction of stress in cylindrical A pressure container is a device designed as a closed container that keeps gases or liquids under pressure that differs considerably from ambient pressure.
Sanal (2000), conducted nonlinear finite element analysis of pressure vessel. In this paper focused on two pressure vessel problem where the large displacement and plastic straining response of the structure is simulated by geometrically and materially nonlinear finite element analysis. In a first case the limit load prediction of imperfect tubes (with ovalized cross sectional shape) under external pressure and discusses the accuracy of the pressure vessel code formula. The second case simulates the large strain cold-deforming process of a pressure vessel made from strain-hardening steel.
Raja et al., (2007) carried out an experiment to study the effect of various parameters such as nozzle diameter, angle of inclination, jet position and jet velocity on mixing time. An increase in the nozzle diameter was found to reduce the mixing time at a given level of power consumption also improve the energy efficiency.
Patil et al. (2016), performed transient finite element analysis of balanced stiffness valve. It is used to determine the time-varying displacements, stresses, strains, and forces in valve parts. Here performance of the Balanced Stiffness Valve, i.e. movement of pressure plates observed. Pressure Plate area is exposing to the fluid flow instantaneously as the supply pressure given to pressure plate. Hence it is essential to examine time dependent dynamic response of the valve. Maximum stress developed within the permissible safety limit also deformation is sufficient for valve performance.
A computational method was carried out by Kong et al., (2014) to simulate the starting transient flow of a vacuum ejector system. The vacuum ejector-diffuser system has been widely used in many applications such as refrigeration systems. The starting transient flows of supersonic vacuum ejector-diffuser system, and its performance characteristics were simulated and analyzed by numerical methods. Primary numerical analysis results show that the chevrons get a positive effect on the vacuum ejector performance: less starting time and

secondary chamber equilibrium pressure are found in chevron transient flow, compared with the convergent nozzle. A CFD method based on transient scheme has been applied to simulate the equilibrium flows and flow dynamics behaviour of the secondary chamber.

Internal mixing chamber twin-fluid nozzles can advantageously replace traditional Y type nozzles to atomize high viscosity fluids did by Barroso et al.,(2014) at industrial power plants are mainly interested on how the resulting SMD is affected by the operating conditions for a given fuel oil a parametric study show that the calculated SMD decreases. When the fuel oil-to-steam pressure ratio is close to1. The lowest SMD is obtained for values of the fuel oil and steam inlet pressures that are very close to the power plant design ones. Hence this nozzle was installed in a power plant.Mixing is one of the common unit operations employed in chemical industries. It is used for blending of liquids, homogenization of mixtures, heat transfer operation, mass transfer operation, prevention of deposition of solid particles, etc.

Kailas et al., (2008), employed CFD to understand the mixing phenomena and to find out the time required for mixing in a jet mixed tank. The 3D simulations are carried out using the CFD package FLUENT 6.2 to generate data and validate them against experimental result.

Khan (2010), studied the analysis of stress distributions in a horizontal pressure vessel and the saddle supports. The results obtained from a 3D finite element analysis show that the stress distribution in different parts of the saddle separately, i.e. wears web, flange and base plates. The effect of changing the load and various geometric parameters was investigated the recommendation are made for the optimal values of ratio of the distance of support from the end of the vessel to the length of the vessel also the ratio of the length of the vessel to the radius of the vessel for minimum stresses both in the pressure vessel and the saddle structure.

Nicolas et al., (2016), did the study to compare the mixing performance of three geometries of HartridgeRoughton mixers with similar dimensions and identical inlet flow rates. Mixing efficiency of three mixers is characterized by a segaration index.

Lu and Hui (2015) have done investigation on mechanical behaviour of cold stretched and cryogenic stretched austenitic stainless steel to develop vessels with thinner shells. The martensite transformations, strength, plasticity properties exhibited by both the methods were compared. FEA based on MISO technique was used to simulate cold stretching and cryogenic stretching. It was observed that by cryogenic stretching the allowable stress of the material can be increased twice than that by cold stretching, promoting reduction in thickness of vessel by 60-75% resulting in light weight vessel.

Kavekar et.al., (2013) have explained about use of composite materials to replace high strength to weight metals for use of pressure vessels in low weight applications such as aerospace and oil and gas. FRP laminates are used which consists of epoxy resins and E-glass fibre plies with suitable coatings. According to the analysis done using ANSYS they have compared steel and the composite for stress and weight. It is found that maximum stress in FRP composite was less than the allowable stress of FRP material. The structural efficiency of FRP vessel was more than steel vessel. Also, weight reduction of 75% was obtained using FRP instead of steel simultaneously removing problem of corrosion of steel.

Raparla and Seshaiah (2012) in their paper have designed multi-layered high pressure vessel and compared it with monoblock vessel. Multilayered pressure vessels are built by wrapping a series of sheets over the core tube. Scope is obtained to select different materials at different layers according to functionality. Inner layer can be made of anticorrosive materials while outer layers can be made of material having high strength. The design is based on ASME Code Section VIII division I. Using multi-layered pressure vessel results in percentage saving of material of 26.02% reducing overall weight of the vessel. With help of FEM software, it is found that the stress variation from inner to outer surface for multilayerd vessel is 12.5% and that for solid vessel is 17.35% resulting in more uniform stress distribution. Thus, the multi-layered pressure vessels are suited for high pressure and high temperature applications.

Deolia and Shaikh (2016) have carried out Finite Element Analysis to estimate burst pressure of mild steel pre ssure vessel using Ramberg-Osgood model. Burst pressure is the pressure at which the vessel bursts or crack and fluid leaks which is undesirable and such pressure must not be exceeded. The burst pressure can be found out numerically using Ramberg-Osgood material curve. The finite element method is used to calculate the burst pressure using Ramberg-Osgood equation and then comparing it with the results obtained from elasto-plastic curve and true stress strain curve. Analysing the results Ramberg-Osgood model showed better correlation with the experimental observations as compared to modified Faupel Formula. Thus the use of FEM can help save time and cost of actual testing.

Sedmaka et.al., (2016) have studied the Elastic-plastic behaviour of welded joints during loading and unloading of pressure vessels. Pressurizing of the model was done in two stages. First the model was loaded to working pressure then held at a lower pressure for 2 hours. After unloading it was tested at total working load or water hammer load. They conclude that higher heat input to the weld zone is better. The HAZ of microalloyed steel has greater resistance to crack during load variations compared to WM. High stress levels for initiation of stable crack growth suggest the possibility that the welded structure can operate safely even in the presence of relatively large surface cracks. The integrity of heterogeneous welded joints is not affected by the presence of surface cracks because overmatching plays a protecting role, which consists in a small plastic deformation of weld metal even at high loads causing fracture of parent metal.

Andrade et.al.,(2015) have done a comparative study on methods to analyse stresses in vessel/nozzle due to external loads. A model of a nozzle without reinforcement is prepared so that comparison can be done by WRC 107, WRC 297 and FEA method. The WRC (Welding Research Council) Bulletin 107 is a parameterized procedure of stress calculation of nozzle in which the input values are dimensionless and the stress results are obtained from curves developed based on experimental data. WRC Bulletin 297 is a supplement to WRC 107 for higher diameter- thickness ratios. The stress values obtained by the three methods were close and are reliable for pressure vessels and nozzles that fit in WRC 107 and 297 procedure.

Gedam and Bhope (2015) have compared the stress distribution for different ends viz. hemispherical, flat circular, standard ellipsoidal and dished shape. The various dimensions of pressure vessel are obtained using analytical procedure. The model is prepared in Pro-E and analyzed using ANSYS. The analysis has been carried out for 2-D Axissymmetric analysis, 3-D horizontal pressure vessel with saddle support and vertical pressure vessel over the stand. From analysis it is observed that the stress generated is less in dished end compared to flat or circular or hemispherical heads, so it is recommended to use ellipsoidal or dished head in vertical as well as horizontal pressure vessels.

For the mixer without a separated mixing chamber, the flow has a double spiral structure. The mixing time is lower than 5 ms for the three mixers and even lower than 1 ms for the mixer with a narrowing end. They recommended the use of a mixer with conical narrowing which has a high mixing efficiency.

Quadir and Redekop, (2008), carried out finite element analysis of a pressurized vessel nozzle interaction with wall thinning damage. Examine the effect of specific tee joint on stress concentration factor of varying wall thinning damage. The largest force found at crotch corner for tee joint with and without wall thinning damage.

Pande (2014), conducted transient dynamic analysis on the tube sheet. It is used as filter in filter tube. Maximum stress and deformation obtained from analysis which is used to calculate infinite fatigue life of tube sheet. Transient dynamic analysis of tube sheet performs to get maximum stress and deformation in tube sheet. Using maximum stress valves, the fatigue life of tube sheet can be calculated.

Shiva Kumar et al., (2018), did the design and analysis of pressure vessel with ASME code. Finite element analysis of the individual model with different materials shows that stress and deformation of valve of modified model is better than actual model. Hence the pressure vessel design of SA516GR65 can be used with modification.

2.2 Inferences from Literature

From the literature study it can observe that the design and analysis of pressure vessel can be studied. In chemical industry storage and distribution of chemical take place in vessel. Due to incoming pressure and temperature creates the deformation in vessel which result high local stress for deeper understanding of stress by combine stress analysis. Maximum stress valve is useful for predicting the failures of pressure vessel.

- The prediction of burst pressure at which crack occurs in pressure vessel by finite element analysis. Due to this crack propagation fatigue life of pressure vessel is reduces. It overcomes by FEA.
- The finite element analysis is generally used to determine the response of a components under the action of different boundary conditions.
- Nonlinear analysis of finite element method is very accurate result than a routine finite element analysis of non-routine problem.
- Combine effect of static and thermal loading conditions on pressure vessel need to be investigate.

3. PROBLEM STATEMENT

Problem statement

In this pressure vessel during operation, maximum operating internal pressure is 1.3 Mpa and operating temperature is 77°C. Hence, it creates continuous deformation and distortion in pressure vessel. A stress induced in vessel beyond the allowable limit, which causing failure of wall and other components. Which cause reduces the fatigue life of vessel. The failure of vessel not only due to the static conditions but it may be due to generation of thermal stresses in vessel material during the operation. Hence, need to validate design calculation with the help of FEM analysis to find out design safety and failure reason of pressure vessel. The analysis results show the failure areas and maximum deformation of vessel with the combine effect of loading conditions in multi-physics analysis. Hence this investigation focusses on "Design and multiphysics stress and vibration analysis of pressure vessel applied in low pressure and medium temperature working conditions."

4. SCOPE OF WORK

Scope

The scope of the work is to validate the manual design calculations with the help of FEM software for safe design and reduce chances of failure during operation. The thermal stress will be relieve by identifying the displacement of sliding saddle.

5. OBJECTIVE

Objective

1. To design pressure vessel for operating pressure 1.3 Mpa and temperature 77°C.
2. To analyse pressure vessel in combine thermal, structural i.e. couple field conditions.
3. To relieve thermal stresses, decide the saddle support movement along longitudinal/ axial direction.

6. METHODOLOGY

Methodology

In this work, as per the discussion with company two cases are presented. The first case focuses on validating the finite element model of a pressure vessel when the right-hand and left-hand saddles were fixed. The stress analysis of vessel was carried out using couple field analysis. The deformation and stress distribution on the pressure vessel using finite element method is compared to analytical method. The second case involves finite element simulation and analysis for the design of a pressure vessel with right hand saddle was free along axial direction and left-hand saddle was fixed in all degree of freedom. The methodology adopted for investigation of design and analysis of pressure vessel. The following steps are involved in the investigation.

Identifying design parameters and material properties as per ASME Section VIII, Div. 2 Part 5.

- Developing finite element model of pressure vessel.
- Static structural analysis of pressure vessel.
- Multi-physics analysis of pressure vessel (Thermo-mechanical) to find the maximum stress and deformation.
- Validating the FEA results against manual design calculations.

7. DESIGN CALCULATION

7.1 Design consideration

A modern chemical process involves more complex and a series of operations which must be run continuously for many months or year. It demands the equipment of exceptional robustness, ingenuity and reliability. A verity of equipment such as pressure vessel is required for storage, handling and processing of chemical. Each equipment performs specific function in some cases it can be suitably modified to perform a different function. Condition such as temperature, pressure etc. under which the equipment is expected to perform.

Industrial horizontal cylindrical pressure vessels are usually supported on twin saddle support, which is used for the purpose of carrying different kinds of products like LPG, petroleum redacts steam and other beverages. Pressure vessels are the most widespread equipment in industrial sector. More precisely vessels are the fundamental component for the industrial importance. Usually, saddles are used to support the horizontal pressure vessel. Apart from the stress due to the internal pressure inside the vessel, saddle has to carry other stresses also such as self-weight of the vessel and other atmospheric conditions. Generally, the pressure vessels are subjected to uniform internal pressure under the effect of liquid contained by it. But due to structure of pressure vessel and loading conditions, it encounters non uniform stresses over its entire structure. So, while we are designing horizontal pressure vessels the design and analysis of its saddle supports are very important step. Saddle stiffness and distances between the saddles have a major effect on the maximum stress induced in the entire structure. The length and diameter of the pressure vessel was chosen from commercial pressure vessel sizing guide.

7.2 Operating condition

The operating condition may be specified as those resulting from the operation during maximum or normal condition, as well as those that exist during starting up or shutting down or during change in loading it is known as transient condition.

7.3 Normal condition

Normal condition includes following things-

- Operating pressure (internal or external) existing during normal operation. The maximum pressure is not more than 10% in excess of the normal valve.

- Operating temperature is decided by contained fluid. The maximum and minimum temperatures have to be specified.
- Influence of environment, including possible corrosion or chemical attack from fluid contained from the atmosphere. Similarly effect of erosion caused by high velocity of flow and effect of irradiation have to be considered.
- External loading such as wind and snow. Other external loading are those resulting from the reaction of piping system attached to the pressure vessel, dead weight of agitator system, pumps, valves, etc., supported by vessel and in general all forms of local loading imposed during service.

7.4 Selection of material

Pressure vessel is major part of equipment used in chemical industry. Selection of material for pressure vessel is depending upon different condition such as pressure, temperature and corrosion effect due to acid and alkalis. The material selection is done based on application point of view, for pressure vessel SA 240-grade 304L, (ASME –Sec-8- div. 2-part D) is mostly used as, 304L is an excellent choice for service in lower than ambient temperature application. It has excellent notch toughness and is used in both pressure vessels and industrial boilers.

In this pressure vessel during operation, maximum operating internal pressure is 1.3 Mpa and operating temperature is 77°C. Hence, it creates continuous deformation and distortion in the pressure vessel. A stress induced in a vessel beyond the allowable limit, which causes failure of the wall and other components. The failure of a vessel not only due to the static conditions but it may be due to generation of thermal stresses in vessel material during the operation. Hence, it is the need to validate design calculation with the help of FEM analysis to find out design safety and failure reasons of pressure vessels. The analysis results show the failure areas and maximum deformation of the vessel with the combined effect of loading conditions in multi-physics analysis. Hence this investigation focuses on, design and multi-physics stress analysis of pressure vessel applied in low pressure and temperature working conditions. The pressure vessel is a major part of equipment used in the chemical industry. The selection of material for 1

Table -1: Mechanical properties for carbon steel material

Properties	Values	Units
Elastic modulus	180000	MPa
Poison’s ratio	0.23	
Tensile strength	485	MPa
Yield strength	262	MPa
Density	7850	Kg/mm^3

Table -2: Typical chemical composition of material

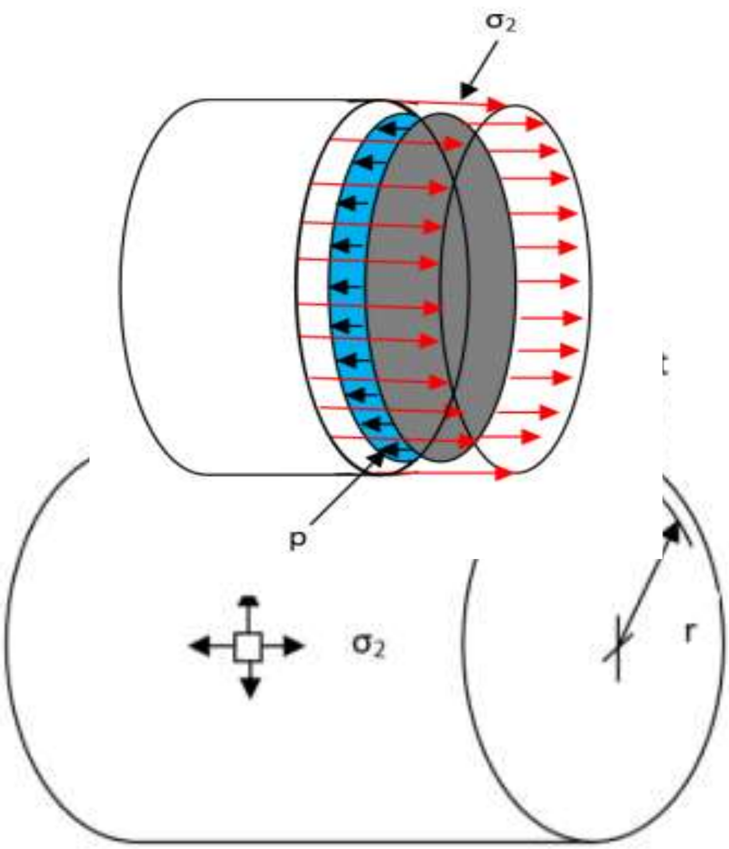
Grade	C	Mn	Si	cr	P	S
SA 240-grade 304L	0.030	2	0.75	18	0.045	0.030

The stress analysis of the pressure vessel was carried out using an analysis tool and compared the results with analytical values. In this work, as per the discussion with company two cases are presented. The first case focuses on validating the finite element model of a pressure vessel when the right-hand and left-hand saddles were fixed. The stress analysis of the vessel was carried out using a couple field analysis. The deformation and stress distribution on the pressure vessel using finite element method is compared to the analytical method. The second case involves finite element simulation and analysis for the design of a pressure vessel with right hand saddle was free along axial direction and left-hand saddle was fixed in all degrees of freedom. The detail design and analysis of the pressure vessel is carried out using two methods i.e., mathematical and finite element methods. The two methods are illustrated in further sections.

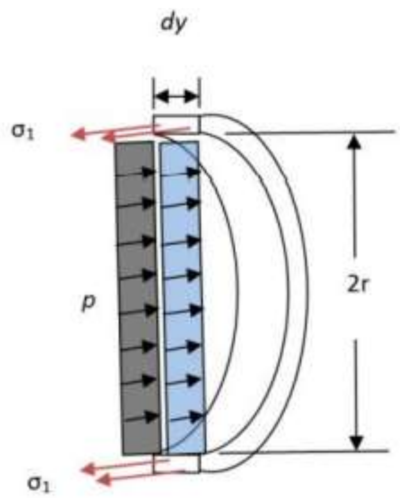
7.5 Mathematical method

In general, analytical approach could be considered as the fundamental tool in solving problems related to engineering mechanics. Therefore, in this study for analytical approach the fundamental equations were used to determine the axial stress and longitudinal stress for the thin-walled pressure vessel respectively. For a cylindrical vessel, a gauge pressure p is developing within the vessel by containing gas or fluid, which is assumed to have negligible weight. Fig. 2 (a,b,c) shows the stress direction, cylindrical pressure vessel subjected to the normal stress $\sigma 1$ in the circumferential or hoop direction and $\sigma 2$ in the longitudinal or axial direction

a) Stress direction



b) Circumferential stress



c) Axial stress

Figure 2: Stress directions

The normal stress σ_1 in the circumferential or hoop direction and σ_2 in the longitudinal or axial direction are represented by Eq. 1 and Eq. 2 respectively.

$\sigma_1 = pr/t$ (1)

$\sigma_2 = pr/2t$ (2) (ASME code)

Eq. 1 and Eq. 2 were used for calculating the stresses and these stresses data were used for the validation of the FEA model.

Analytical calculations are carried out using following input design parameters as per the customer requirement as shown in Table 3.

Table -3 : Pressure vessel design parameters

Design pressure	1.3 Mpa
Design temperature	77°C
Vessel thickness	16 mm
Inner radius	1500 mm

Allowable stress calculation-
Allowable stress = yield strength / factor of safety
= 262/1.5
= 174 Mpa
Actual allowable stress = allowable stress X 0.85 (duty factor)
= 174 X 0.85
= 148 Mpa

- Design of cylindrical shell
  Allowable stress = 148 Mpa

(According to ASME Section-VIII Division-I, UG36)
Thickness of shell, t = 16 mm
Circumferential stress = σ_1= pr/t = (1.3 x 1500) / 16
= 121.875 Mpa
Longitudinal stress= σ_2 = pr/2t = (1.3 x 1500) / (2 x 16)
= 60.9375 Mpa

8. FINITE ELEMENT METHOD

The finite element method (FEM) is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. An unsophisticated description of the FE method is that it involves cutting a structure into several elements (pieces of structure), describing the behavior of each element in a simple way, then reconnecting elements at nodes as if nodes were pins or drops of glue that hold elements together (Fig. 3). This process results in a set of simultaneous algebraic equations. In stress analysis these equations are equilibrium equations of the nodes. There may be several hundred or several thousand such equations, which mean that computer implementation is mandatory (Patel et al.,2013).

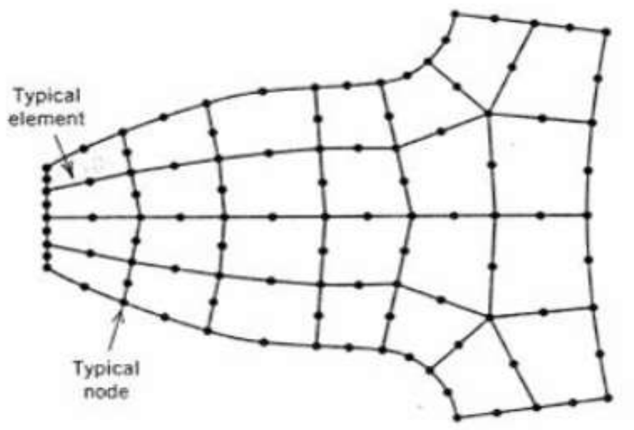


Figure - 3: Discretization principle (Patel et al.,2013)

8.1 Why FEM

The Rayleigh-Ritz method and potential energy approach are now of only academic interest. For a big problem, it is difficult to deal with a polynomial having as many coefficients as the number of DOF. FEM is a better generalization of these methods and extends beyond the discrete structures. Rayleigh-Ritz method of choosing a polynomial for displacement field and evaluating the coefficients for minimum potential energy is used in FEM, at the individual element level to obtain element stiffness matrix (representing load-displacement relations) and assembled to analyse the structure (Narasaiha, L. 2009). (Narasaiha, L. 2009).

8.2 Origin of FEM

One of the most common examples of engineering analysis is finite element analysis or FEA. FEA is one of the most commonly used and powerful features of the CAD software. To carry out the analysis of object by using FEA, the object is divided into finite number of small elements of shapes like rectangular or triangular. These objects form the interconnected network of the concentrated nodes. Analysis is being carried out by using ANSYS software. Finite Element Analysis is a mathematical representation of a physical system comprising apart/assembly (model), material properties, and applicable boundary conditions (collectively referred to as (pre-processing), the solution of that mathematical representation (solving), and the study of results of that solution (post processing). Finite element analysis (FEA) involves solution of engineering problems using computers. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be easily analyzed. FEA software provides a complete solution including deflections, stresses, reactions, etc (Narasaiha, L. 2009)..

8.3 Principle of FEM

In FEM, the entire structure is analysed without using assumptions about the degree of fixity at the joints of members and hence better estimation of stresses in the members was possible. This method generates a large set of simultaneous equations, representing load-displacement relationships. In FEM, actual component is replaced by a simplified model, identified by a finite number of elements connected at common points called nodes, with an assumed behaviour or response of each element to the set of applied loads, and evaluating the unknown field variable (displacement, temperature) at these finite number of points.

Use of proper techniques for modeling a structure, the boundary conditions and, the limitations of the procedure, are very crucial. Engineering structures, e.g., bridge, aircraft wing, high-rise buildings, mechanical component etc., are examples of complex structures that are extremely difficult to analyze by classical theory. But FEA technique facilitates an easier and a more accurate analysis. In this technique the structure is divided into very small but finite size elements (hence the name Finite Element Analysis) and that method to divide the structure into small element is called as Discretization Method (Narasaiha, L. 2009)..

FEA is used to determine following structural properties;

- Static displacement and static stress.
- Natural frequencies and mode shapes.
- Forced harmonic response amplitude and dynamic stress
- Transient dynamic response and transient stress.
- Random forced response, random dynamic stress.

8.3.1 FEA requires three steps for finding deflections and stresses in a structure.

- Pre-process or modelling the structure.
- Analysis.
- Post processing.

A brief description of each of these steps is explained below.

Step1: Pre-processing

The pre-processing involves modeling of the structure, selection of element type, meshing of the body, material properties, applying the boundary conditions, and applying the loads. The FEA model consists of several elements that collectively represent the entire structure. In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called "mesh"

Step 2: Analysis

The processing involves generation of element stiffness matrices and global stiffness matrix, solution of simultaneous equation, determination of nodal displacement, stresses and strains. The form of the individual equations, as well as the structural equation is always,

$\{F\} = [K]\{u\}$

Where,

$\{F\}$ = External force matrix.

$[K]$ = Global stiffness matrix

$\{u\}$ = Displacement matrix.

The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis

Step 3: Post processing

This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behavior of the structure.

Structural analysis is the most common application of the Finite Element Method (FEM). The structural analysis is a mathematical algorithm process by which the response of a structure to specified loads and actions is determined. This response is measured by determining the internal forces or stress resultants and displacements or deformations throughout the structure.

8.3.2 Types of Structural Analysis

- Static Analysis
  This analysis used to determine displacements, stresses, etc. under static loading conditions. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.
- Modal Analysis
  It is used to calculate the natural frequencies and mode shapes of a structure.
- Harmonic Analysis
  It is used to determine the response of a structure to harmonically time-varying loads.
- Transient Dynamic Analysis
  It is used to determine the response of a structure to arbitrarily time-varying loads.
- Spectrum Analysis
  An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum.
- Buckling Analysis
  Used to calculate the buckling loads and determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analyses are possible.

In structural analysis using structural loads to produce a variety of structural results. By default, 3D structural simulations are performed. Static used when loads are constant for individual sets of results. Typical application include determining safety factors, stresses and deformation for a body or assembly under structural loading.

8.3.3 Types of supports

- Fixed supports
- Displacement
- Remote displacement
- Frictionless support
- Compression only support
- Cylindrical support
- Elastic support.

8.3.4 Types of loads

- Pressure load

Pressure is uniform and acts normal to a surface at all locations on the surface. A positive pressure acts into the surface, compressing the solid body. If you select multiple surfaces when defining the pressure, the same pressure value gets applied to all selected surfaces.

- Displacement

It requires one or more flat or curved surfaces to displace relative to their original location by one or more components of a displacement vector in the world coordinate system or local coordinate system, if applied.

Zero Y component. No part of the surface can move, rotate or deform in the Y direction. Blank (undefined) X and Z components. The surface is free to move, rotate, and deform in the XZ plane. Use multiple select to apply a displacement load to more than one surface.

Note

- Entering a zero for component prevents deformation in that direction.
- Entering a blank for a component allows free deformation in that direction.
- Using multiple displacements on the same surface and on surfaces having share edges.

Other types of loads are

- Hydrostatic Pressure
- Bearing load
- Bolt pretension
- Moment

- Inertial loads

8.3.5 Element selection

For analysis software needs all three dimensions. It cannot make calculations unless and until geometry is defined completely. Geometry is classified based on dominant dimension as follows and hence the element type is selected accordingly
1-D: One dimension is very large compared to the other two
2-D: Two dimensions are very large compared to the third one
3-D: All three dimensions are comparable
It is mandatory to choose proper method and element size for the meshing. As the heat exchanger thickness is very small as compared to the other two dimensions it is desired to go for 2D meshing. Therefore, a shell element was used for discretization process. Meshing has done manually the method used is quadrilateral dominant method.

Figure 4 shows node rectangular shell elements with the shell mid-surface nodal points

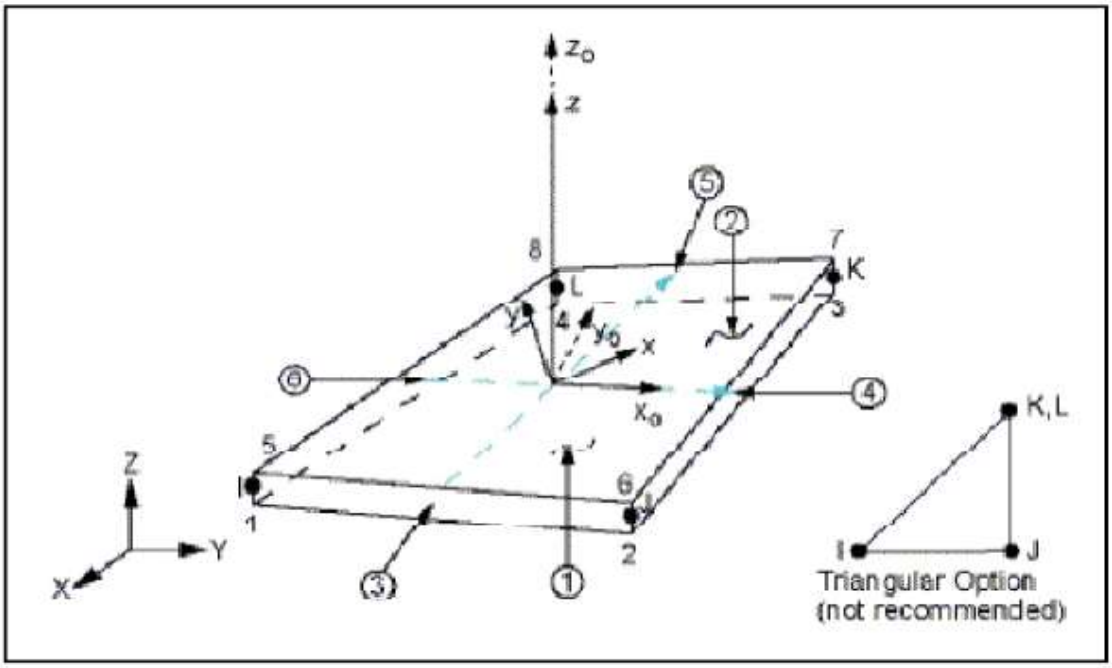


Figure - 4: Node rectangular shell elements with the shell mid-surface nodal points

The 4-node rectangular shell element makes it easier to obtain the stresses in the direction of the plate edges for this nonlinear plate analysis. This shell element can be used with elastic-isotropic, plastic-bilinear and plastic multi-linear material models. It can also be used in a large displacement/small strain problem. This type of element is suitable for the present application. To predict material behaviour under multi-axial loading, a yield criterion, that indicates for which combination of stress components transition from elastic to plastic deformations occur, should be used. The applicable yield criterion for metal plasticity is the von-misses yield criterion. The von-misses yield criterion has been interpreted physically as implying that plastic flow occurs when shear strain energy exceeds a critical value. The von-misses criterion is often used to estimate the yielding of ductile materials. Also, this criterion is largely based on the experimental observation that most polycrystalline metals are isotropic. Carbon Steel is an isotropic and ductile material.

- **Shell181**

SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a 4- node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as a filler element in mesh generation. SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.

- **Quadrilateral Dominant Method**

In ANSYS workbench, for surface model's solver directly selects Quad Dominant method of meshing. While meshing a multimode part that contains a mix of line bodies and surface bodies, all surface bodies and all line bodies that share edges with surface bodies will be meshed with the selected surface mesh method. Any remaining line bodies (where only vertices are shared with surface bodies) will always be meshed with the Quadrilateral Dominant mesh method. If you select the Quadrilateral Dominant method (default), the body is free quad meshed. The Quadrilateral Dominant mesh method includes the following settings:
Free Face Mesh Type - Available for most analyses and can be set to either Quad/Tri (default) or All Quad. Element midsize nodes - The global element mid-side nodes option allows user to control whether meshes are to be created with mid-side nodes (quadratic elements) or without mid-side nodes (linear elements). Reducing the number of mid-side nodes reduces the number of degrees of freedom. Choices for the global element mid-side nodes option include Program Controlled, Dropped, and Kept. Examples are presented below. The heavy vertical line in each graphic represents the body boundary. Program Controlled - Program Controlled is the default. For surface bodies and beam models, Program Controlled is identical to the Dropped option described below. For solid bodies and 2-D models, Program Controlled is identical to the Kept option described below.

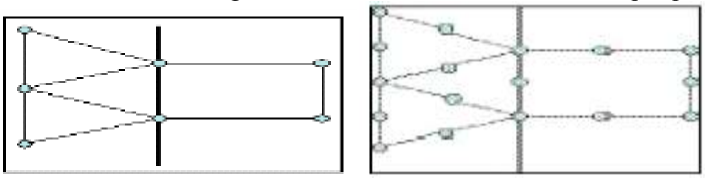


Figure - 5 : Triangular elements with dropped & kept options

Dropped - The Dropped option removes mid-side nodes on all elements. Examples shown below are for a solid body.
Kept - The Kept option retains mid-side nodes on elements created in the part or body. All elements will have mid-side nodes.

- Element quality check

Different quality parameters are the measure of how far a given element deviates from the ideal shape. Following are the quality parameters to be checked.
9 Element quality
10 Aspect ratio
11 Max corner angle
12 Jacobian ratio
1) Element quality: It is the criteria to measure deviation from an ideal shape. It should range between 0 to 1
2) Aspect ratio: It is the ratio of largest dimension of an element to smallest dimension between adjacent sides. It should range between 1 to 5.
3) Max corner angle: It is the measure of corner angle of an element. It should range between 90 to 140 for a quad element.
4) Jacobian ratio: It is the ratio of change of local co-ordinate to change of global co-ordinates. It should range between 1 to 5.
Fig.8 a) shows the CAD geometry of pressure vessel.

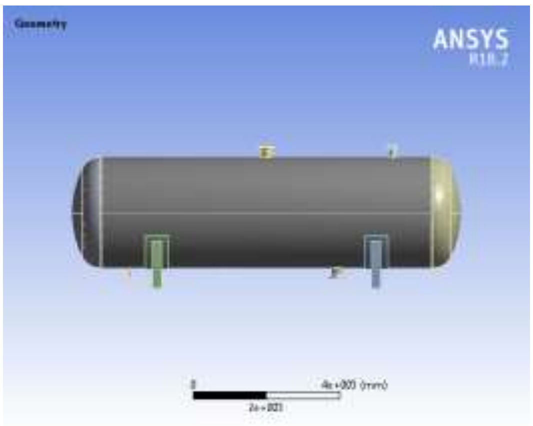


Figure - 6 : CAD Geometry

REFERENCES

1. Adithya, M., & Patnaik, M. M. M. (2013). Finite element analysis of horizontal reactor pressure vessel supported on saddles. International Journal of Innovative Research in Science, Engineering and Technology, 2(7), 3213-3220.
2. Pendbhaje, A., Gaikwad, M., Deshmukh, N., & Patil, R. (2014). Design and analysis of pressure vessel. International Journal of Innovative Research in Science and Technology, 2(3), 28-34.
3. American Society of Mechanical Engineers. (1998). ASME boiler & pressure vessel code section VIII division 1. New York: American Society of Mechanical Engineers. (Cited in Zengliang Gao, et al., 2005).
4. American Society of Mechanical Engineers. (1998). ASME section II part D. New York: American Society of Mechanical Engineers. (Cited in Zengliang Gao, et al., 2005).
5. Deolia, P., & Shaikh, F. A. (2016). Finite element analysis to estimate burst pressure of mild steel pressure vessel using Ramberg–Osgood model. Perspectives in Science, 8, 733-735.
6. Gedam, A. C., & Bhope, D. V. (2015). Stress analysis of pressure vessel with different end connections. International Journal of Mechanical Engineering, 3(11), 19-27.
7. Gupta, S. R., Desai, A., & Vora, C. P. (2014). Optimize nozzle location for minimization of stress in pressure vessel. IJIRST-International Journal for Innovative Research in Science and Technology, 1(1).
8. Huda, Z., & Ajani, M. H. (2015). Evaluation of longitudinal and hoop stresses and a critical study of factor of safety (FoS) in design of a glass-fiber pressure vessel. International Journal of Civil, Environmental, Structural, Construction and Architectural Engineering, 9(1), 39-42.
9. Patil, K., & Patil, V. (2016). Analysis of balance stiffness valve by using transient analysis method. International Journal on Recent and Innovation Trends in Computing and Communication, 7, 145-151.
10. Kavekar, M. M., Khatawate, M. V. H., & Patil, M. G. V. (2013). Weight reduction of pressure vessel using FRP composite material. International Journal of Mechanical Engineering and Technology (IJMET), 4(4), 300-310.
11. Khan, S. M. (2010). Stress distributions in a horizontal pressure vessel and the saddle supports. International Journal of Pressure Vessels and Piping, 87(5), 239-244.
12. Kong, F., & Kim, H. D. (2014). Starting transient simulation of a vacuum ejector-diffuser system under Chevron effects. International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics.
13. Lu, Y. Q., & Hui, H. (2015). Investigation on mechanical behaviors of cold stretched and cryogenic stretched austenitic stainless steel pressure vessels. Procedia Engineering, 130, 628-637.
14. Mundhe, N. D., & Utpat, A. A. (2013). Analysis of cracked cylindrical pressure vessel by using experimental approach. International Journal of Latest Trends in Engineering and Technology (IJLTET), 2(3), 155-160.
15. Di Patrizio, N., Bagnaro, M., Gaunand, A., Hochepied, J., Horbez, D., & Pitiot, P. (2016). Hydrodynamics and mixing performance of Hartridge Roughton mixers: Influence of the mixing chamber design. Chemical Engineering Journal, 283, 375-387.
16. Nabhani, F., Ladokun, T., & Askari, V. (2012). Reduction of stresses in cylindrical pressure vessels using finite element analysis. INTECH Open Access Publisher.