ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY

SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING

GRADUATE PROGRAM IN RAILWAY ENGINEERING

Stress and Fatigue Analysis of Cylindrical Oil Tank Wagon with Elliptical Head

A ThesisSubmitted to the School of GraduateStudies of AddisAbaba institute of Technology in Partial Fulfillment of the Requirement for the Degree of Masters of Science in Mechanical Engineering (Railway Engineering)

PREPARED BY:

AbdourahmanIsmail Bodleh

ADVISOR:

TolossaDeberie (Mr.)

November 2017

ADDIS ABABA, ETHIOPIA

ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY

SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING

GRADUATE PROGRAM IN RAILWAY ENGINEERING

Stress and Fatigue Analysis of Cylindrical Oil Tank Wagon with Elliptical Head

By:

Abdourahman Ismail Bodleh

Approved By Board of Examiners

TolossaDeberie (Mr) _______________ ______________

Advisor: **signature** **date**

_______________ _______________ _______________

Internal examiner: **signature** **date**

_______________ _______________ _______________

External examiner: **signature** **date**

Daniel Tilahun (Dr.) ______________ _______________

Head, railway center: **signature** **date**

DECLARATION

I, undersigned, declarethatthisthesisismy original work and has not been presentedfor a degree in this or anyotheruniversities, and all sources of materialsused for thethesiswork have been fullyacknowledged.

Abdourahman Ismail Bodleh ________________ ___________________

Name Signature Date

This thesis has been submitted for examinationwithmyapproval as a universityadvisor.
Mr:Tolossa Deberie ________________ ___________________

Name Signature Date

ACKNOWLEDGMENT

First, I would like to extend my heartfelt thanks to my advisor, Mr.TollossaDeberie (PHD Candidate), for getting me started on the path which this paper has taken. Indeed his inspiration and guidance throughout the paper work starting from its conception and design and stress analysis of cylindrical oil tank wagon have been invaluable.

My appreciation also goes to the Ethiopian Railways Corporation for its financial support and Addis Ababa Institute of Technology for its unreserved provision of resources relevant information and advice for in for undertaking this thesis.

I would also like to thank the staff of the EthiopiaRailways Corporation, for providing me with the necessary data.

Finally, I am indebted to my family especially to myfather and mother and my sister mouna and also to my wife Kadra Raaliya Abdillahi Moussa and to all my friends for their understanding and encouragement throughout my studies. They have all been atremendous source of hope and support throughout this work.

ABSTRACT

A cylindrical tank wagon is a type of railroad freight car designed to transport chemicals, petroleum products, and other bulk liquids like oil.Itis a welded metal tank of cylindrical shape, placedhorizontally.

Thisstudyaims analyze and identify the stress and fatiguedeformation of cylindricaloil tank wagon to minimize the deformation. The modelling of the cylindrical tank structure along with its simplified supporting frame of the wagons isdone by using the Solidworks 2013 software to show the geometry and the relevant component.

After accomplishing the modellingon solidworks, itwasimportedinto the ANSYS v17.2 to analyze the stress, deformation and also FEMFAT was done to analysis the fatigue life of the tank. The meshing type for this research is Fine size meshing. All the properties material are applied. From the analysis of stress and fatigue which has been carried out through the deformation of the cylindrical tank in ANSYS and FEMFAT softwares, the material analysis was conducted and it yielded simulated result.

The analysisincludes total deformation, stress, strain, and fatigue life, safety factorwasdoneusing ANSYS and also endurance limit and survival propability was done by FEMFAT softwares. The analysis of the simulation carried out throught changing the material has yielded the maximum deformations, standing at 0.0065458 Max carbone steel and at 0.003612 Max AISI A656 High StrengthAlloySteel, therefore the deformationof cylindrical tank reduce by 44.81%. FEMFAT software reveals that the selected material have greater endurance limit and survival probability, which mean the new material will be more resistant. So thisrecommendsthe application of the new materiel AISI A656 High StrengthAlloySteel for the analysis of cylindrical tank wagon material.

Key Words: Stress, Deformation, Cylindrical Tank, Materials, ANSYS and FEMFAT softwares.

TABLE OF CONTENT

LIST OF TABLE

LIST OF FIGURE

List of Symbols

σ_L=Longitudinal or axial stresses

σ_c=Circumferential or ring stresses

σ_r=Radial stresses

P =Tank pressure

W=wallthickness

D=Tank diameter or diameter of head

T=Thickness of shell and head

F=Force of the tank

ρ =Density of carbone steel

A =Cross section area

L=Shell length

m_1=Mass of tanker

m_2=Mass of end covers& Baffle plates

M_T=Total masse of tank

g =Acceleration due to gravity

v =Poisson's ratio

E =Young'smodulus

ε_L= longitudinal or axial strain

ε_c=Circumferentialstrain

ε_v=Volumetricstrain

Kf = Fatigue Strengthreduction factor

Ke,k = Fatigue Penalty Factor

Salt, k =Alternating stress amplitude

ΔSP, k =Equivalent Stress

N=Number of cycle

X and Y = Unknown

$C_1 C_2 .... C_{11}$ = Coefficient of fatigue curves.

ET= Young'smodulus for material.

S_a =Intensified Stress Amplitude

CHAPTER: ONE

INTRODUCTION

1.1. Background

A tank car is a type of railroad freight car whose body is made up of a horizontal tank (shell) designed to transport chemicals, petroleum products, and other bulk liquids and compressed gases. The cylindrical body or shell is sealed at its ends by means of hemispherical or ellipsoidal heads.Tank wagons are designed are taking consideration the properties of dangerousgoods,so for the transportation of such goods, tanks are equipped with special devices for performing loading/unloading operations and for providing the security of transportation.The barrel of a tank wagon may be in use for the carriage of cargo without overpressure (petroleum, food, chemicals, etc.) or pressurized (liquefied gases).To protect the metal of the barrel from causing corrosion on the carried substances, special internal coatings or additives of corrosion inhibitors are applied to the cargo.

In 1865, Amos and James Densmore designed a more cost-effective way to transport oil by rail. The brothers invented and built the Densmore Tank Car, a special railroad car that could haul 80-90 barrels (nearly 3,400 gallons) of oil**.** Most oil was actually shipped in wooden barrels and oil tank wagons which were not common until the later 1920's, when people started building specialised rail-connected reception facilities. Until early 1970's chemical tank wagons used plain mild steel tank barrels and they were confined to a single traffic. By the early 1980's there was increasing shift toward stainless steel (or 'staybright') tanks and they were washed out and used for a different wagon [2].

Fahy and Tiernan (2001) model an ISO tank container was statically and dynamically in use for rail, road and sea conditions using ANSYS, with the objective of producing a more efficient and safer design,where the areas of high stress in the tank and support structure were identified. Years later, Domagala and Lisowski (2011) indicate that fluid structure interaction and simulation represent a very efficient tool for design mobile tanks, especially for dynamic loading. Shortly Zhang (2012) indicates that the difference of the liquid volume in tank has great influence on the structural stress and strain**[**2].

But in the case of cylindrical tank, it slides from side to side where very little weight is moving, butit does not move far from the centre line so the wagon remains stable.

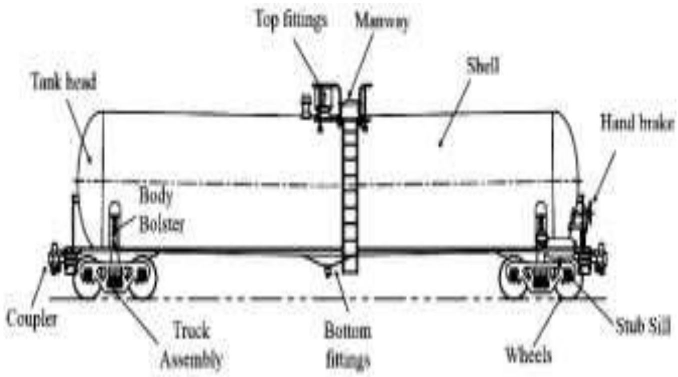


Figure 1: Principal design elements of a regular Tank-Car.

The tank container concept was designed by Bob Fosseyan, engineer, who worked for Williams Fairclough in London.In 1964, he made a swap body tank for a combined transport by truck and train, but this tank was not yet constructed in accordance with ISO standards. In 1966, the first commercial production occurred and one year later the first tank container according to ISO standards was developed.The first mass-produced tank containers were purchased by Trafpak, a part ofPakhoed.In 1969, the ISOTANK, was registered as a name; and the first ISO container tank in the world gotthe Lloyds Register and the UK DoT Hazardous Goods Department approvals for international transport. George Lambert was head of the Division and Design; and their first order was for 10 off insulated and electrically heated units for hazardous substances and 10 for none- hazardous goods. In the early 1970s, the tank container evolved into its current form and the production was also well underway. In the early days, the production took place in Europe [3].

Other researcher continued their researches on interaction of oil and structure based on finite elements method by applying two-dimensional (Jacobsan, 1949) and three-dimensional models (Jaiswal et al, 2003) and this issue has been considered by many researchers. On the other hands, different methods have been used based on attitude of the designers, with two general classes of static and dynamic analysis**.**

1.1.1 Different Types of tank wagons

A. Milk cars

A milk car is a specialized type of tank car designed to carry raw milk between farms, creameries and processing plants. Milk is now commonly chilled, before loading, and transported in a glass-lined tank car. Such tank cars are often used as "Food service use only".

Figure 2: milk cars tank [1]

B. Liquid hydrogen tank car

Tank cars of this type are designed to carry cryogenic liquid hydrogen (LH_2). North American cars are classified as DOT113, AAR204W, and AAR204XT.

Figure 3: liquid hydrogen tank car [1].

C. Pickle cars

A pickle car was a specialized type of tank car designed to carry pickles. This car consisted of several wooden or metal parts (typically three or four) and was often roofed. Pickles which are preserved in salt brine were loaded through hatches in the roof.

D. Tank containers

A tank container, also known as ISO tank, is a specialized type of container designed to carry bulk liquids, such as chemicals, liquid hydrogen gases and food grade products. Both hazardous and non- hazardous products can be shipped to tank containers. A standard tank container is 20 feet (6.10 m) long, 8 feet (2.44 m) high and 8 feet (2.44 m) wide. The tank, which is made from stainless steel, is held within a box-shaped frame with the same shape as an intermodal container. This allows it to be carried on multiple modes of transport, such as truck, rail and ship**.**

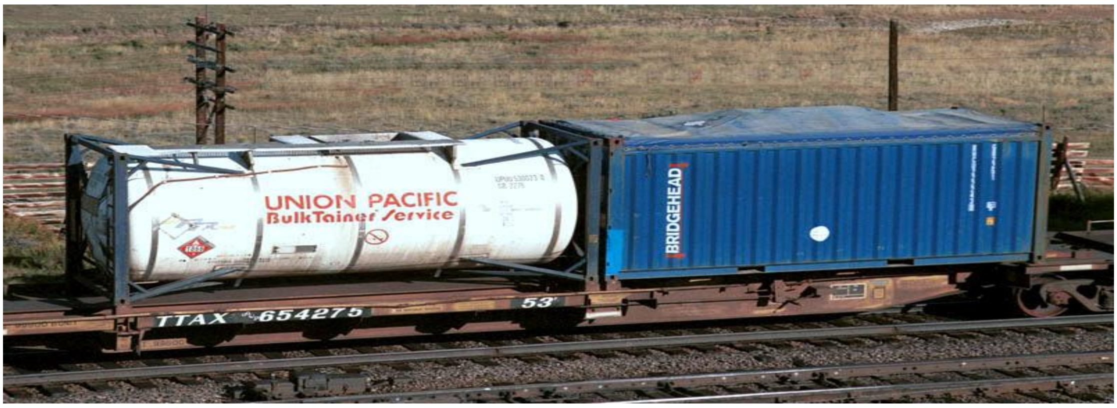


Figure 4: Tank container [2].

E. Torpedo cars

A **torpedo car** or **bottle car** is a type of railroad car used in steel mills to haul moltenpig iron from the blast furnace to begin the process of primary steelmaking.The thermally insulated vessel is mounted and designed to endure extremely high temperatures, as well as keeping the metal in a molten state over extended periods of time. The vessel can be pivoted along its longitudinal axis to empty the pig iron into a ladle. The name is derived from the vessel's resemblance to a torpedo.

Figure 5: Torpedo cars [2]

F. Vinegar cars

A vinegar car is a specialized type of tank car designed to transport vinegar. The largest such car built was built by the Morrison Railway Supply Corporation in 1968. The car's underframe included all of modern facets of freight car design including roller bearing trucks and cushioning devices built by Freight Master, while the tank that rode on it, was made of douglas fir, and had a capacity of 17,100 US gallons (65 m^3; 14,200 imp gal). The car, which has been called 'the largest wooden tank car ever built', taking 18 months to build. Vinegar is now moved in ordinary tank cars lined with glass, plastic, or alloy steel.

Figure 6:vinegar cars [2].

1.2. Statement of problem

In conducting the research and analysis of the new transportation oil tank wagon, the cylindrical oil tank wagon can be used to study problems related to the stress, deformation and fatigue life of the tank. In Ethiopia the rail vehicles are new technology and it is decided to transporting of oils and petroleum fluids from port to the capital city, Addis Ababa by train wagons. Therefore it is very important to show the stress ; deformation and fatigue life of cylindrical tank wagon during static condition. With the growing numerical technique this research has been decided to be conducted the numerical analysis to determine the longitudinal and circumferential stress for the tank wagon.The safety issue of these tank wagon transportation systemsis the required activity to reduce maximuim deformation of the tank wagon.So, in this thesis, it is mostly focused the analysis of freight tank wagon to analyze the pressure vessel analysis of thin wall cylinder and this research result has been determined the survival probability, endurance limit , stress and deformation of cylindrical oil tank and finally compare with the conventional material.

1.3. Objective of the study

1.3.1. General objective

The major objective of this research is stress and fatigue analysis ofcylindrical oil tank wagon with elliptical head based on finite element method.

1.3.2. Specific objective

The specific objectives are listed below, by implication, with the results expected from this thesis.

1. To study stress analysis of cylindrical oil tank wagon to determine the longitudinal and circumferential stress for the tank.
2. Modeling and simulation of cylindrical tank body using softwares SOLIDWORKS, ANSYS and FEMFAT.
3. To analyze the static analysis of cylindrical oil tank wagon to determine the stress, deformation and fatigue life.
4. After analyzing the currently used material in comparison with this materials and finally selectof the appropriate materials for the tank wagon.

1.4. Scope and limitation of the study

1.4.1 The scope of the research

This study is limited to:

- construction of 3D model of the tank wagon using software SOLIDWORKS;
- to conduct stress pressurized analysis of cylindrical Oil Tank wagon based on longitudinal , Circumferentiel and hoop stress of the tank;
- pressure vessel analysis using ANSYS software,
- Fatigue analysis using FEMFAT software,
- interpreting of the result and conclusion and finally,
- Togive recommandation to the result obtained from simulation.

1.4.2. Limitation of the research

Some of the limitations encountered during the conduct of this research work are ignoring the dynamic effect and for being theoretical, without practical testing in the work shop and also limited time and Resource.

1.5. Methodology

1. **Litteraure review:** reading journal and previous researches have been carried out in this project and also sources are taken from different manufacturer catalogue by visiting a reliable websit.
2. **Data collection:** The design standards and collecting the specifications of the body components have been taken from Ethiopian Railway Corporation. Browsing different published research articles and journals from the internet and visiting ERC.
3. **Modeling of the structural Tank Wagon Body:** A software namely solid works is used for modeling of structural oil tank wagon body.
4. **Research design:** This thesis work employs observational type of method to achieve its objectives, taking into considerations previous research work, and all the parameters as presented in this research simulation, Rail standard specifications are also dealot with.

CHAPTER: TWO

2. LITERATURE REVIEW

2.1. Description of Tank Wagon

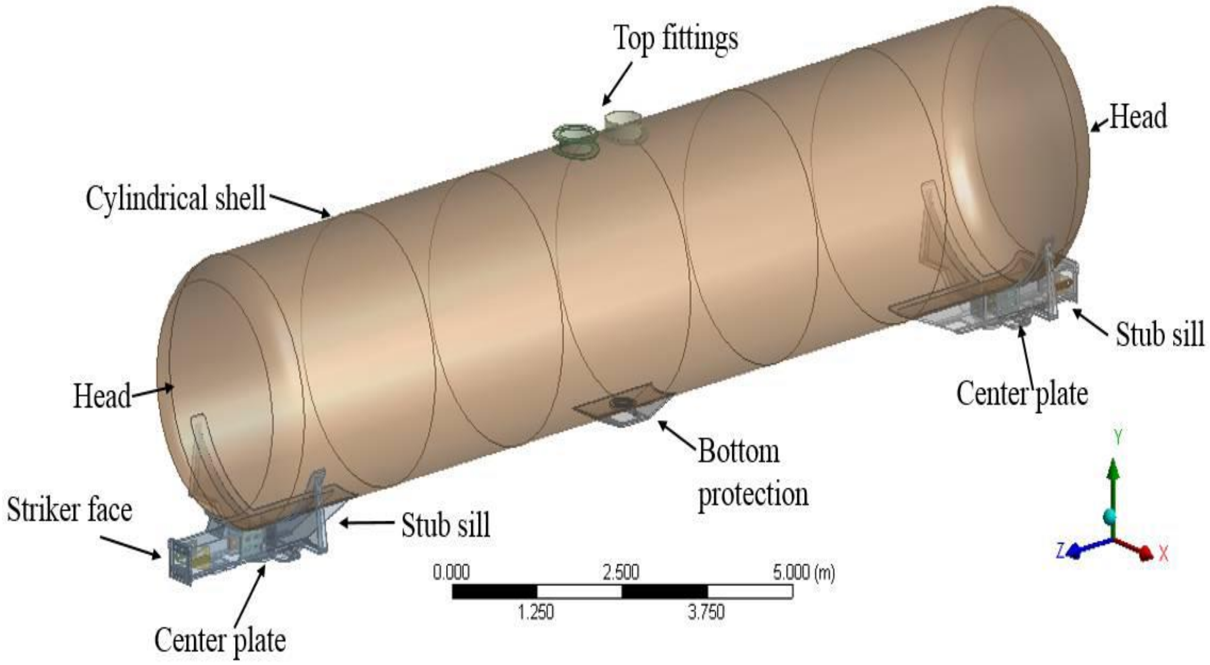


Figure 7: Principal Parts of the Performed Model [7].

Top fitting: Tank cars have top fitting protection including structural steel housing with the safety valve contained within the house.

Shell: The cylindrical section of a tank car without head.

Head: One of the ellipsoidal ends of a tank car.

Body bolster: The structural membres at each end of a carbody that support the car on its track assembles.

Bottom fitting: A plugged and flanged oopening in the bottom of a tank to facilitate cleaning of a tank car does not have a bottom outlet.

Stub sill tank car: Atank car design with draft sills at each end of the tank instead of a continuous center sill that utilizes the tank as part of the car structural.

Truck : The assembly of wheels ,axles, bearing ,spring , side beraing ; side frame and bolster that support each end of a rail car on enable its to move on the rail.

2.1.1. The Tank shell Deformation and Its Origins

The surface of the tank shell was greatly corrugated. Its shape was deformed during the construction period, and also later, throughout almost forty years of its service life. The deformation generated during the production period includes mainly angular refraction at vertical and horizontal but welds of the tank shell [3].Another cause of deformation during the construction period is the imperfection of calculation of the perimeter of the first shell ring at the bottom rim of the tank. The next cause of new deformations or an increase in the existing deformation is the process of irregular tank subsidence into the sand foundations. It is connected with the consolidation degree of the sand foundations and with the value of concentrated load of the tank shell's dead weight. The existing shell deformation state was recreated on the basis of geometric measurements, which were taken on 40 plumb-lines located uniformly on the tank's perimeter [3].

2.2. Design Parameter Of Pressure Vessel

The following are design parameters of pressure vessel

1. design Pressure
2. allowable stress, and
3. corrosion allowance

2.2.1. Design Pressure

In the pressure vessels, three terms related to pressure are commonly used

- Maximum Working pressure is the maximum pressure to which the pressure vessel is subjected.
- Design pressure is the pressure for which the pressure vessel designed
- Hydrostatic test pressure is the pressure at which the vessel is tested. The pressure vessel is finally tested by the hydrostatic test before it is put into operation.

In this research, the working pressure of the cylindrical tank is 1.5MPa and the factor considered is the force (F) and area of the tank (A).

2.2.2. Allowable Stress

As per the ASME Code, the allowable stress is based on the ultimate tensile strength with a factor of safety of 3 and 4 respectively.
As per the IS Code, the following stress is obtained on the yield strength with a factor of safety of 4.
Therefore, Allowable stresses are:

$$\sigma_{all} = \frac{Sut}{3}$$

Where, σall = allowable tensile stress for the pressure vessel, N/mm^2

Sut= ultimate tensile strength for the pressure vessel material,N/mm^2

2.2.3 Corrosion Allowance

The walls of the pressure vessel are subjected to thinning due to corrosion which reduces the life of the pressure vessel. The corrosion in pressure vessel is due to the following reasons:

- Chemical attack by reagents on the inner wall surface of the vessel.
- Rusting due to atmospheric air and moisture.
- High temperature oxidation.
- Erosion due to flow of reagent over the wall surface at high velocities [12].

2.3. Stresses in pressure vessels

Stress is the internal resistance or counterforce of a material to the distorting effects of an external force or load, which depends on the direction of applied load as well as on the plane it acts. At a given plane, there are both normal and shear stresses. However, there are planes within a structural component subjected to mechanical or thermal loads that contain no shear stress. Such planes are principal planes, the directions normal to those planes are principal directions and the stresses are principal stresses. For a general three-dimensional stress state there are always three principal planes along with the principal stresses.

Different types of stresses are as follows:

1. Circumferential stress,
2. Longitudinal stress,
3. Radial stress.

According to D. R. Moss (2004) stresses are generally categorised as primary, secondary or peak stresses. Primary stresses are stresses due to pressure (internal or external), mechanical loads and wind which can result in the rupture or total collapse of a pressure vessel, they are the most hazardous. Secondary stresses on the other hand are strain-induced stresses, and can be developed at the junction of major components of a pressure vessel (e.g. radialloads on nozzles) because of stresses caused by relenting load or differential thermalexpansion. While Peak stresses are the maximum stress concentration point in addition tothe primary and secondary stresses present in a region. Peak stresses are only significant infatigue conditions and are the sources of fatigue cracks, which are applicable to membrane, bending and shear stresses (Rao. K. R.2002).When a thin-walled cylinder is subjected to internal pressure, three mutually perpendicular principal stresses will be set up in the cylinder material, namely the circumferential or hoop stress, the radial stress and the longitudinal stress, (Sharma.S.C .2010). Provided that the ratioof thickness to inside diameter of the cylinder is less than 1/20**,** it is reasonably accurate toassume that the hoop and longitudinal stresses are constant across the wall thickness, and that the magnitude of the radial stress set up is so small in comparison with the hoop andlongitudinal stresses that it can be neglected. This is obviously an approximation since, inpractice, it will vary from zero at the outside surface to a value equal to the internal pressureat the inside surface [17].

2.4. Reviewing research works

Boyko .A: Studies the Analysis of damage element of barrel where fatigue cracks appear during exploitation time. To substaintiale heightened level of stress in local areas of the barrel additional calculation were performed in accordance with elaborated role of loading. Result shows that barrel damage is due to high level of pressure in the barrel [4].

Y.Y. SEMIN AND M.N. Kulyapin: Study the stress-deformed state in dents area on the wall of verticall steel tanks. During design work strength and stability calculations of vertical steel tanks were performed using momentless theory of shells. The shape of tank wall is considered as an ideal cylinder. It is evident that in dents area stressed-deformed state of the wall differs from stress-deformed state in ideal cylinder and can not be calculated using momentless theory of shells. As a case study for calculation of stress-deformed state vertical steel tank with floating roof has been selected VSTFR-10000 a capacity of 10000 m^3. Calculation of stressed-deformed state of wall has been performed using finite element method by modeling

VST wall with ANSYS. During calculations the maximum dent tongue (Fmax) was set from 7,8 mm up to 273 mm with a step of 7,8 mm; all in all 35 dents have been createdwith equal width, height, shape, same location on the tank wall and different as far as dent tongues were concerned. As a result of inspection a considerable number of dents occurred inboth erection and operation [6].

V.N. Skopinsky and A.B. Smetankin: describes the structural model and stress analysis of nozzle connections in ellipsoidal heads subjected to external loadings. They used Timoshenko shell theory and the finite element method. The features of the structural model of ellipsoid-cylinder shell intersections, numerical procedure and SAIS special-purpose computer program were discussed. A parametric study of the effects of geometric parameters on the maximum effective stresses in the ellipsoid-cylinder intersections under loading was performed. The results of the stress analysis and parametric study of the nozzle connections are presented. [8].

Prof. Vishal V. Saidpatil 1, Prof. Arun S. Thakare:study Design and Weigh optimization of Pressure Vessel Due to Thickness Using Finite Element Analysis, The aim of this paper was to carry out detailed design & analysis of pressure vessel used in boiler for optimum thickness, temperature distribution and dynamic behavior using Finite element analysis software.Finite Element Method is a mathematical technique used to carry out the stress analysis. In this method the solid model of the component is subdivided into smaller elements. Constraints and loads are applied to the model at specified locations. The research paper was carried out involves design of a cylindrical pressure vessel to sustain 5 bar pressure and determine the wall thickness required for the vessel limiting the maximum shear stress. Geometrical and finite element model of Pressure vessel is created using CAD CAE tools. Geometrical model is created on CATIA V5R19 and finite element modeling is done using Hypermesh. ANSYS is used as a solver Weight optimization of pressure vessel due to thickness using FEA.[9].

Ibrahim et al.: studyStress Analysis of Thin-Walled Pressure Vessels; this paperdiscusses the stresses developed in thin-walled pressure vessels. Pressure vessels (cylindrical or spherical) are designed to hold gases or liquids at a pressure substantially higher than the ambient pressure. Equations of static equilibrium along with the free body diagrams will be used to determine the normal stresses $\sigma_1$in the circumferential or hoop direction and $\sigma_2$in the longitudinal or axial direction. Pressure vessels can be any shape, but shapes made of sections of spheres and cylinders are usually employed. A common design is a cylinder with end caps called heads. Head shapes are frequently hemispherical. The normal stress in the walls of the container is proportional to the pressure and radius of the vessel and inversely proportional to the thickness of the walls. As a general rule, pressure vessels are considered to be thin-walled when the ratio of radius *r* to wall thickness *t* is greater than 10. Pressure vessels fail when the stress state in the wall exceeds some failure criterion. Therefore pressure vessels are designed to have a thickness proportional to the radius of tank and the pressure of the tank and inversely proportional to the maximum allowed normal stress of the particular material used in the walls of the container. Cracked or damaged vessels can result in leakage or rupture failures. Potential health and safety hazards of leaking vessels include poisonings, suffocations, fires, and explosion hazards. Rupture failures can be much more catastrophic and can cause considerable damage to life and property [10].

Apurva R. Pendbhaje,and Mahesh Gaikwad : carried out design And Analysis Of Pressure vessel ; Pressure vessels are usually spherical or cylindrical with dome end. The cylindrical vessels are generally preferred because they present simple manufacturing problem and make better use of the available space. Tanks, vessel and pipelines which carry, store or receive fluids are called pressure vessel. A pressure vessel is defined as a container with a pressure differential between inside and outside. The inside pressure is usually higher than the out-side. The fluid inside the vessel may undergo a change in state as in the case of steam boiler or may combine with other reagent as in the case of chemical reactor. Pressure vessel often has a combination of high pressure together with high temperature and in some cases flammable fluids or highly radioactive material. Because of such hazards it is imperative that the design be such that no leakage can occur. In addition vessel has to be designed carefully to cope with the operating temperature and pressure. They get result that Vessel failures can be grouped into Three major categories, which describe why a vessel failure occurs :

- Material- Improper selection of material; defects in material.
- Design- Incorrect design data; inaccurate or incorrect design methods; inadequate shop testing.
- Fabrication- Poor quality control; improper or insufficient fabrication procedures including welding [11].

V. V. Wadkar, S.S. Malgave, D.D. Patil , H.S. Bhore , P. P. Gavade : The main objective of this study is to design and analyze the features of pressure vessels. Various parameters of Solid Pressure Vessel are designed and checked according to the principles specified in American Society of Mechanical Engineers (A.S.M.E) Sec VIII Division 1. The stresses developed in Solid wall pressure vessel and Head of pressure vessel is analyzed by using ANSYS, a versatile Finite Element Package. The theoretical values and ANSYS values are compared for both solid wall and Head of pressure vessels. In general, pressure vessels designed in accordance with the ASME Code, Section VIII, Division 1, are designed by rules and do not require a detailed evaluation of all stresses. It is recognized that high localized and secondary bending stresses may exist but are allowed for by use of a higher safety factor and design rules for details. It is required, however, that all loadings (the forces applied to a vessel or its structural attachments) must be considered. Theoretical calculated values by using different formulae are very close to that of the values obtained from ANSYS. This indicates that analysis is done with a valid model of pressure vessels. Analysis of cylindrical pressure vessel with hemispherical head type is performed and maximum equivalent stresses found by ANSYS are compared with theoretical values. It is concluded that smaller values of equivalent stresses are appearing in pressure vessel with hemispherical heads, and equivalent stress distribution is advantageous too in that case of head geometry [12].

R.C. Carbonari et al.: discuss shape optimization of axisymmetric pressure vessel with objective of minimizing Von Mises mechanical stress from head to nozzle. They found that entire independent optimization of nozzle and heads lead to undesirable results and hence they have considered the entire model from head to nozzle for which they used commercial software ANSYS and MATLAB. They used Shell 208 as element type. They concluded that it is important to entire vessel for shape optimization, allowing the method to minimize different stress level between head and nozzle by considering the coupling cylindrical region [13].

PuneetDeolia, Prof. F.A. Shaikh: studied Burst Pressure Analysis of a Pressure Vessel, Pressure vessels are used for various industrial applications. To ensure safe operation of pressure vessel it is important to know its burst pressure. Results obtained by analytical approach are not reliable and experimental calculation is costly as well as hazardous. Thus finite element approach can be used as an alternative to predict the burst pressure which ensures the safety, reduce cost of testing and increase accuracy as compared to analytical formulas Researchers used different approaches to evaluate burst pressure namely analytical, Experimental and Numerical. They suggested avoiding use of analytical approach to avoid unnecessary conservatism. Amongst various formulas available, Faupel formula gives most reliable results [14].

Zainul Huda, Muhammad Hani Ajani:studyEvaluation of Longitudinal and hoop Stresses and a Critical Study of Factor of Safety (FoS) in Design of a Glass-Fiber Pressure Vessel, The design, manufacture, and operation of thin-walled pressure vessels must be based on maximum safe operating pressure and an adequate factor of safety (FoS). The research paper first reports experimental evaluation of longitudinal and hoops stresses based on working pressure as well as maximum pressure; and then includes a critical study of factor of safety (FoS) in the design of a glass fiber pressure vessel. Experimental work involved the use of measuring instruments and the readings from pressure gauges. The control devices and measuring tools were used to observe working fluid pressure and the geometric design parameters of the water-filter tank. The longitudinal stress and hoop stress in the tank were evaluated to be in the range of **1.25 – 3.73**MPa. The FoS was computed to be 55; which was found to be more than safe to operate the machine because the FoS value is much higher than required value of FoS=4 [15].

SourabhLawate, B. B. Deshmukh: studyhead design in pressure vessels which is a challenging task. For the pressure vessels, different types of heads or ends can be provided. In this work, a comparative study of different types of pressure vessel heads is discussed. A finite element method based on software ANSYS is used to observe the stresses in these heads. Axi-symmetric behaviour of elements is used to reduce the modeling and also analysis time. Quadrilateral elements with mid-side nodes are used for meshing the models. Comparison of stresses in these types of heads is done to study the differences in stresses and considering the forming cost and the stresses developed, torispherical head is found better than elliptical head and hemispherical heads. From the present work, the result it is observed that stresses and deformation in hemispherical heads is the lowest. Maximum Von Mises Stresses in elliptical and torispherical heads are observed 2.5 times (150.67 % more) & 3.58 times (258.22 % more) respectively as compared to Maximum Von Mises stresses in hemispherical head. Maximum deformations observed in elliptical and torispherical heads are 9.26 times (826.83 % more) and 9 times (801.96 % more) respectively as compared to deformation in hemispherical head. Observed deformations in elliptical and torispherical dish ends are in close agreement. As forming cost of torispherical heads is less than elliptical heads and hemispherical heads, it is recommended for the present case (P<10bar), though the stresses are slightly on higher side than elliptical head [18].

KirtikumarTamboli :studied Fatigue Analysis of Pressure Vessel by FEA Techniques, This requires a fatigue analysis including thermal and stress analyses. In general the cylindrical shell is made of uniform thickness which is determined by the maximum circumferential stress due to the internal pressure. The life of a vessel under cyclic service is related to the intensity of the stress and the number of cycles it is exposed to. The fatigue life curves used under ASME VIII-2 to calculate the permitted cycle life of a vessel are based on a large factor of safety compared with actual cycle life curves. We are using VIII-2 fatigue methods to calculate an allowable number of operating cycles with a factor of safety, not to predict the cycle life of the vessel which normally will be larger Hence a finite element technique has been selected for analysis purpose. Fatigue analysis is carried out for entire equipment for specified regeneration cycles and found fatigue life more than required cycles. There are different types of commercial FEM software’s available in market. ANSYS FEM software is one of the most popular commercial software is used for finite element analysis of vessel. Stress Analysis by finite element method is obviously best choice. The objective of analysis was to check fatigue life of 2000Ltr Air Receiver for cyclic pressure service and impact loading service in accordance with ASME Section VIII, Div-2 Part 5 Ed. 2013. To study the stress levels, finite element based stress analysis is carried out. The result show that the Maximum deformation of 1.4409 mm occurs near the dish ends and the maximum fatigue damage fraction observed was less than unity as required by code. Accordingly we conclude that all evaluation points for fatigue are within allowable limits specified by code. [19].

PrashantNanavare and AbhayUtpat: study fatigue Analysis of 6300 Liters Pressure Vessel by Using Cyclic Service.This study has carried out fatigue analysis during arbitrary transients and a result are applied to a finite element model along with the pressure to calculate the stress and deformation values estimate number of cycles as per ASME standard. Fatigue analysis was done to calculate allowable useful life cycle of the vessel against the designed cycle. High pressure is developed in pressure vessel so pressure vessel has to withstand several forces developed due to internal pressure implying that selection of pressure vessel is most critical for safetypurpose. Accordingly we conclude that all evaluation points for fatigue are within allowable limits specified by code. The maximum fatigue damage fraction observed was less than unity as required by code [20].

2.5. Summaries of the reviewed literatures:

From various literature reviews above mentioned are investigated the past research that has been done in many areas related to this work. Most of the literatures listed in this thesis focus on the analysis of pressure vessels of the tank by analyse the shell and head shape.Also some of literature reviews are related the analysis thin-walled pressure vessel and fatigue Analysis of Pressure Vessel by FEA Techniques, and the methods are used, most of them are used ANSYS software based finite element method. Therefore, it is important to determine the fatigue and stress deformation of cylindrical tank with different materials when the pressure is applied on the tank.Then this research does the analysis the currently used materials for ERC which is the carbon steel and also AISI A656 High Strength Alloy Steel material and how they reacts the pressure applied to resist the stress deformation in the cylindrical tank wagon and finally suggest the better material.

CHAPTER THREE

3. Material, Dimension, Method, Analytical Model

3.1. Materials

The GW2 Tank Wagon is provided for the railway in the gauge of 1435 mm in Ethiopia, mainly for shipments of gasoline, kerosene and diesel, in the mode of top loading and top unloading. The maintenance of the railway wagon can mainly be divided into two kinds, one is the car repair in work, and another is the car repair in depot.

3.1.1. Fluid Wagon

The study was conducted considering the dimension and the parameters, model GW2 ER1004 tank wagon attached with two bogies of two axles each, and the bogies are attached with the tank wagon. Technical data reached ERC and the photo show as it reached at sabeta station.

3.2. Dimension

3.2.1. The Material Selection

The carbon steel material used in pressure application covers a very wide range of mechanical properties. Carbon steel materials are listed in the American Society of Mechanical Engineers (ASME) boiler and pressure vessel. Most of the higher strength material has very limited application in power plants. , Accordingly, the materials covered in this report will be limited to those with a specified minimum tensile strength less than 550MPa.Carbon steel is used in the United States and throughout the world for nearly all of the same reason: its cost, properties ease of fabrication, availability, weldability, and so on.

High strength alloy steel is a type of alloy steel that provides better mechanical properties or greater resistance to corrosion than carbon steel. High strength alloy steel varies from other steel in that it is not made to meet a specific chemical composition but rather than to specific mechanical properties. It has carbon content between 0.05-0.25% to retain formability and weldability.These element are intended to alter the microstructure of carbon steel which is usually a ferrite-pearite aggregate, to produce a very fine dispersion of alloy carbides in an almost pure ferrite matrix, increasing yield strength by 50% .High strength alloy steel usually require 25 to 30% more power to form as compared to carbon steel.

Table 1: Properties of the materials of tank wagon[22].

Mechanical property	Carbone steel	AISI A656 High Strenght alloy steel
Young's modulus ,	206.85 GPa	250GPa
Poisson's ratio ,	0.3	0.3
Yield strength	344.74 MPas	652 MPas
Ultimate strength	558.48 MPas	755 MPas
Density	7860 Kg/m^3	7980 Kg/m^3

3.2.2. Specification of tank wagon

Table 2: Rail Tank specification [22].

Tare weight	23.8 t
carrying capacity	70.0t
Minimum radius of curvature negotiable	145m
Design speed	100km /h

3.2.3. Principal Characteristics Of The Model

Table 3: principal geometric characteristics of the model [22].

Shell and head length	10.769m
Diameter of the head	3.05 m
Thickness of shell and head	0.0127 m
Total volume of the tank	65.4 $\mathbf{m^3}$
Effective volume of tank	62.8 $\mathbf{m^3}$
Operating pressure of tank	1.5 MPa
Bogie center distance	8.050 m
Maximum width of wagon	3.320 m
Maximum height of wagon	4.495 m
Fixed wheel base	1.830 m
Wheel diameter	0.840 m

3.3. Method

The procedural method for this study to reach to the simulation and by then the graphical, tabulated as well as the animation results is shown as the following flow chart:

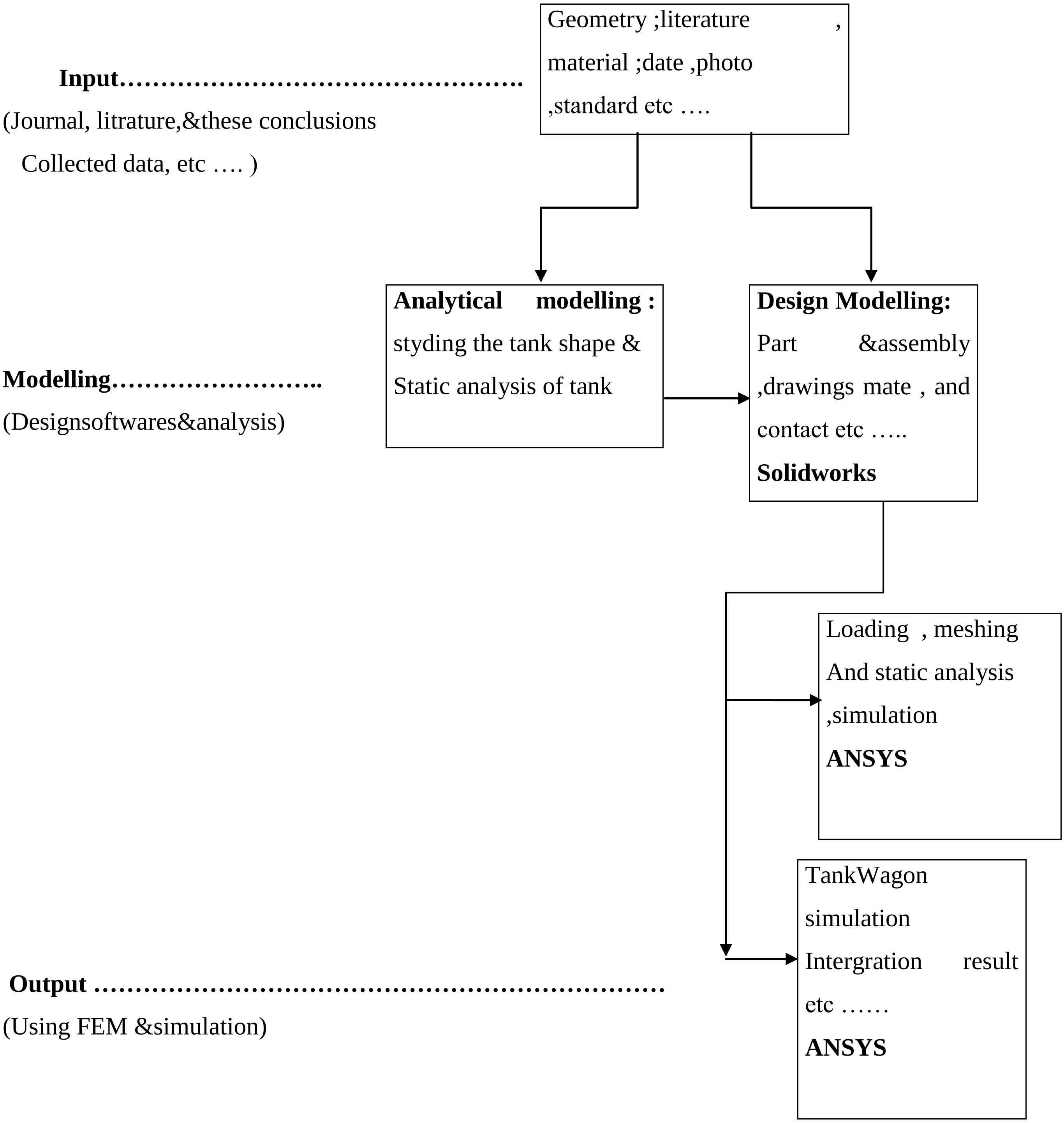


Figure 8: Flow-chart of basic methodology

3.4. Analytical model

3.4.1. Geometric Model

The design of railroad tank cars is subject to structural and performance requirements and constrained by weight. According to the geometric configuration of the wagons, the level of transported liquid may directly influence the structural mechanical behavior of the units. The physical model consist a cylindrical fuel tank wagon of a moving freight train, which is partially filled with diesel/Naphtha (ρ =830 kg/m3, μ = 0.042 kg/m-s). The dimensions of the tank are of 10.769 m length and 2.6 m diameter. The Diesel occupies three different fill-level of the total volume of the space in the cylindrical tank. The detail various dimensions of the physical model of fuel tank wagon as per of GW2 type China made Ethiopian wagon are given in Fig 3.2[22].

Figure 9: Real appearance of GW2 ER 1001 oil tank freigh (ERC)

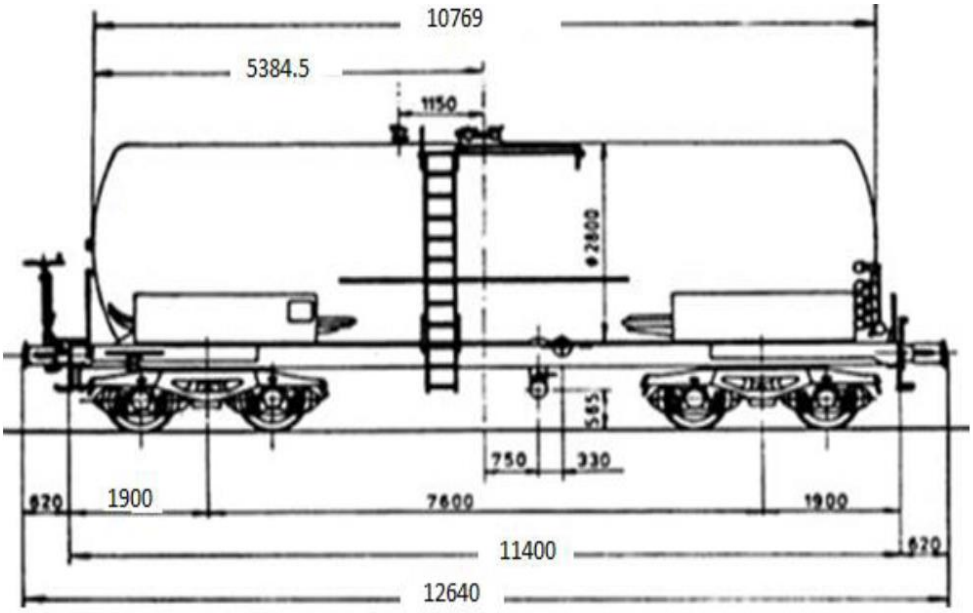


Figure 10: Schematic appearance and dimension of GW2 ER1001 Oil Tank freight

3.5. Numerical Model

3.5.1. Deformations and Stresses of Cylindrical Tank Wagon

Depending on the type of material, size, geometry of an object and forces applied, various types of deformation may result:

1. **ElasticDeformation**

This type of deformation is reversible. Once the forces are no longer applied, the object returns to its original shape. Linear elastic deformation is governed by Hooke's law, which states: $\sigma=E\times\varepsilon$

Whereσ is the stress, ε indicates the resulting strain and E is a material constant called Young´s modulus. This relationship only applies in the elastic range and indicates that the slope of the stress vs. strain curve can be used to find Young's modulus. The elastic range ends when the material reaches its yield strength. At this point plastic deformation begins.

2. The effect of oilattemperature

Temperature changes affect volume measurements in sever always. The dimensions of the tank, and the density and level of the liquid it contains vary with temperature, and the ambient temperature is different from the temperature in the process tank.Temperature variations can affect soundwaves by affecting sound velocity. The liquid level sense disbased on the time required for aplane acoustic wave to travel from the top of the tank to the liquid surface and to return. Typically, the time for a large process tank to cool from 55° C to 30° C isfrom 15 to 48 hours depending on the size of the tank and the ambient conditions in the cell. In order to observe a change in temperature, the test liquid should be preheated to about 60° C, and the temperature range covered was from 47 to 23° C ; so in this research paper the temperatureis 42° C that means there is not effect of oil at this temperature[23].

3. Stresses

The tank stress depends on:

i) tank pressure P
ii) wall thickness w
iii) tank diameter D
iv) tank L/D
v) Heating pattern (fill level, fire contact area, etc.)

4. Thin wall cylinder

The stresses in thin-walled cylinders have been wide studied, due to the huge amount of applications that they have, such as boiler shells, pressure tanks, pipes… etc

The stress produced in the longitudinal direction is $\sigma_L$andin the circumferential direction is σ_C. These are called longitudinal and circumferential stresses respectively. The later is also called the hoop stress.

Generally three types of stresses are developed in pressure shells:

- Longitudinal or axial stresses σ_L
- Circumferential or ring stresses σ_C

- Radial stresses σ_{r}

Two types of analysis are commonly applied to pressure vessels. The most common method is based on a simple mechanics approach and is applicable to "thin wall" pressure vessels which by definition have a ratio of inner radius, r, to wall thickness, t, of $r/t \geq 10$.

4.1) Longitudinal or axial stress (σ_L)

So along as the wall thickness is small compared to the diameter and and then the force trying to split it due to the pressure is :

$F = pA = p\frac{\pi D^2}{4}$ ……………….............. (1.1)

So long as the material holds then the force is balanced by the stress in the wall. The force due to the stress is :

$F = \sigma_L$ multiplied by the area of the metal $=\sigma_L \pi D t$………………(1.2)

Equation 1.1 and 1.2 we have

$\sigma_L = \frac{pD}{4t}$ …………………………….…….(1.3)

The mass in the cylindrical tanker is, according to Bernoulli, handled as a Body which is accelerated by the force difference acting on the cross sections at both ends of the cylinder. The force difference depends on the total pressure difference of both reservoirs consisting of local pressure difference, height level difference and height level velocity difference.

Mass of tanker (m_1) = Density (ρ) x Volume (V_1)

$= \rho$(carbone steel) x $V(volume of the tank)$

$=7860$ kg /m^3$\times(\pi \times D \times L \times t)$

$=7860\times \pi \times 2.6 \times 9 \times 0.0127$

$= 7338.24 Kg$ … … … … … … … … … … ..(14)

Mass of end covers & Baffle plates (m_2) = 6(Density (ρ) x plate Volume (V_2))

$= 6\{ \rho$(carbone steel) $\times$ [(W) (D) (t)]}

$= 6\{7860\times (3)\times (2.6) \times (0.0127)\}$

$=4671.66$ kg ……………….(1.5)

From (a) & (b), Total mass of the cylindrical tanker with elliptical head =7338.24+4671.66 =12009.9Kg

$F=M_T\times g = 13639.97 \times 9.81 = 117817.2\ N$

The operating pressure is 1.5MPa and the diameter is 2.6m and also the thickness is 0.0127m.

From equation (1.3) $\sigma_L = \frac{pD}{4t} = \frac{1.5\times2.6}{4\times0.0127} = 76.7MPas$

$\sigma_L = 76.7MPas$

4.2) Circumferential or ring stresses σ_c

Now considered the forces trying to split the cylinder along a length. The force due to the pressure is:

$F = pA = pLD$……………………............ (1.6)

So long as the material holds then the force is balanced by the stress in the material . The force due to the stress is :

$F = \sigma_c$ multiplied by the area of the metal= $\sigma_c 2Lt$…………... (1.7)

Equation 1.6and 1.7 we have

$\sigma_c = \frac{pD}{2t}$ ……………………………..(1.8)

$\sigma_c = \frac{pD}{2t} = \frac{1.5\times2.6}{2\times0.0127} = 153.5\ MPas$

$\sigma_c = 153.5\ MPas$

4.3) Radial stresses σ_r

The radial stresses are normal to the curved plane of the isolated element. In thin-walled cylinder theory, they are normally not considered,because they are negligible and smallcompared to the other two stresses.

4.4) Strain

There are two stresses perpendicular to each other, the axial ($\boldsymbol{\sigma L}$) and circumferential ($\boldsymbol{\sigma c}$) stresses, as explained before.

From basic stress and strain theory the corresponding axial strain is:

$$\varepsilon_L = \frac{1}{E}(\sigma_L - v\sigma_c)$$

E is the modulus elasticity and $\boldsymbol{v}$ is poisson's ratio. Substituting $\sigma_L = pD/2t$ we have

$\varepsilon_L = \frac{\Delta L}{L} = \frac{1}{E}(\frac{pD}{4t} - v\frac{pD}{2t}) = \frac{pD}{4tE}(1-2v)$................ .(1.9)

$\varepsilon_L = \frac{1.5\times 2.6}{4\times 0.0127\times 206.85\times 10^9}(1-2\times 0.3) = 1.48\times 10^{-10}$

$\varepsilon_L = 1.48\times 10^{-10}$

The circumferential strain may be defined as follows.

$\varepsilon_c =$ Change circumference /Original circumference

$$\varepsilon_c = \frac{\pi(D+\Delta D)-\pi D}{\pi D} = \frac{\Delta D}{D}$$

The circumferential strain is the same strain based on diameter , in other words the diameter strain.

From basic stress and strain theory, the corresponding circumferential strain is:

$\varepsilon_c = \frac{1}{E}(\sigma_c - v\sigma_L)$

Substituting $\sigma_L = pD/4t$ and $\sigma_c = pD/2t$ we have

$\varepsilon_c = \frac{\Delta D}{D} = \frac{1}{E}(\frac{pD}{2t} - v\frac{pD}{4t}) = \frac{pD}{4tE}(2-v)$...................(2.0)

$\varepsilon_c = \frac{1.5\times 2.6}{4\times 0.0127\times 206\times 10^9}(2-0.3) = 2.516\times 10^{-10}$

$\varepsilon_c = 2.516\times 10^{-10}$

Now we may deduce the change in diameter, length and volume.

Original cross sectional area of cylinder= $A_1 = \pi D^2/4$

Original length $= L_1$ Original volume$= V_1 = A_1 L_1 = (\pi D^2/4)(L_1)$

New cross sectional area = $A_2 = (\pi\Delta D + \Delta D)^2$

New length=L$_2$ =L+ΔL

New volume= $V_2 = A_2$L$_2$= {$(\pi\Delta D + \Delta D)^2$} L+$\Delta L$

Change in volume= $\Delta V = V_2 -$ V$_1$

Volumetric strain= $\varepsilon_v = \frac{\Delta V}{V1}$

Then, the volumetric strain can be written as

$$\boldsymbol{\varepsilon_v = \frac{\Delta V}{V} = \frac{\left(\frac{\pi(D+\Delta D)^2}{4}\right)(L_1+\Delta L) - \left(\frac{\pi D^2}{4}\right)L_1}{\left(\frac{\pi D^2}{4}\right)L_1}}$$

Dividing out, clearing brackets and ignoring the product of two small terms, this reduces to

$\varepsilon_v = \frac{\Delta L}{L1} + 2\frac{\Delta D}{D} = \varepsilon_c + 2\varepsilon_L$ …………………….. (2.1)

Substituting the equations 1.9 and 2.0 for the axial and circumferential stresses into this, leads to :

$\varepsilon_v = \frac{pD}{4tE}(5-4v)$ …………………………………...(2.2)

$\varepsilon_v = \frac{1.5\times2.6}{4\times0.0127\times206\times10^9}(5-4\times0.3) = 1.416\times10^{-9}$

$\varepsilon_v = 1.41\times10^{-9}$

3.5.2. Fatigue Analysis

In materials science, fatigue is the weakening of a material caused by repeateadly applied load, and it is the progressives and localized structure damage that occurs when a material is subjected to cyclic loading. The nominal maximum stress values that cause such damage may be much less than the strength of the material typically quoted as the ultimate tensile stress limit or the yield stress limit. Fatigue occurs when a material is subjected to loading and unloading.TheAmerican society for testing and materials defines fatigue lifes, N$_f$As the number of stress cycles of a specified character that a specimen sustains before failure of a specified number occure. Fatigue is Essentials intended to provide a user friendly tool for conducting Stress-Life analysis either using classical stress calculations, while many parts may work well initially, they often fail in service due to fatigue failure caused by repeated cyclic loading in practice, the purpose of this analysis is to show that this pressure equipment is capable of with standing the operation for its intended life cycles. Pressure and temperature is changed for operating condition. Due to this stress ranges are induced by the pressure and temperature variations, fatigue crack can be initiated at the discontinuity where fatigue strength is very weak.

A. Fatigue Simulation Processusing fatigue simulation program FEMFAT

A simulation tool for fatigue assessment is FEMFAT, developed by the Austrian company Magna Steyr. It is well-established software used by various companies in the automotive and engineering industry. All analyses within this Thesis were carried out with FEMFAT MAX version 5.0 and whenever it is written FEMFAT in this paper. In the first step the material data, i.e. the FE-model, has to be imported into FEMFAT. The material properties should be put it to have some result.. It is possible to evaluate 2D and 3D elements but in this Thesis the analyses are limited to 3D elements. By putting the material properties as a input will lead to different result.

B. Experimentation Fatigue Life Calculation

According to design data, for Cyclic Pressure Service of 0 MPa to 3.5 MPa design. The effective alternating stress amplitude for the K^{th}cycle is:

$$S_{alt,k} = \frac{K_f . K_e . \Delta S_{p,k}}{2} = \frac{1.2 \times 1 \times 427.82\text{MPa}}{2} = 256.692\text{N/mm}^2$$

Where K_f =Fatigue Strength reduction factor =1,2

K_e =Fatigue Penalty Factor =1 (since ΔSp,k<ΔSPS; i.e. 427.82MPa < 558 Mpa).

$\Delta S_{p,k} = 427.82\text{MPa}$

To calculate the design numbers of cycles following formula is used: N= 10^X

Where , X= $\frac{C_1+C_3Y^1+C_5Y^2+C_7Y^3+C_9Y^4+C_{11}Y^5}{1+C_2+C_4Y^2+C_6Y^3+C_8Y^4+C_{10}Y^5}$

Y= $\frac{S_a}{C_{us}} \times \frac{E_{FC}}{E_T}$

The coefficients C1, C2... are calculated from table – Coefficient of fatigue curves. Carbon, Low Alloy, Series 4XX, High Alloy Steels, and High Tensile Strength Steels for Temperature not exceeding 371℃ , $\sigma_{uts} \leq 558$ MPa (80 ksi).

Table 4: Coefficient of fatigue curves (ASME Section VIII, Division 2, Part 3)

Coefficients C_1	$48 \leq S_a \leq 214$ (MPa)	$48 \leq S_a \leq 3999$ (MPa)
1	2.254510E+00	7.999502E+00
2	-4.642236E-01	5.832491E-02
3	-8.312745E-01	1.500851E-01
4	8.634660E-02	1.273659E-04
5	2.020834E-1	-5.263661E-05
6	-6.940535E-03	0.0
7	-2.079726E-02	0.0
8	2.010235E-04	0.0
9	7.137717E-04	0.0

Note： E_{FC} = 195E3MPa (28.3E3ksi)

Compute Primary Membrane Stress [S]:

$$\frac{P}{E\times\ln\left(\frac{2\times t+d}{d}\right)} = \frac{0.40}{1\times\ln\left(\frac{2\times12.7+2600}{2600}\right)} = 4.368 \text{ MPas}$$

Sample calculation for the Intensified Stress Amplitude [S_a]: $\frac{S\times3.3}{2} = \frac{4.368\times3.3}{2}$ =7.20 MPas

Stress Factor used to compute X [Y]: $= \frac{S_a}{C_{us}} \times \frac{E_{FC}}{E_T} = \frac{7.20}{1} \times \frac{195E3}{206.85E9} = 6.81\times 10^6$

Compute from Equation :$\frac{C_1+C_3Y^1+C_5Y^2+C_7Y^3+C_9Y^4+C_{11}Y^5}{1+C_2+C_4Y^2+C_6Y^3+C_8Y^4+C_{10}Y^5} = 6.145$

Compute the Number of Cycles from Equation : N= $10^X = 10^{6.145} = 1.3910^6$ Cycles

CHAPTER: FOUR

Modelling and Simulation of Cylindrical Tank

4.1. Modelling software

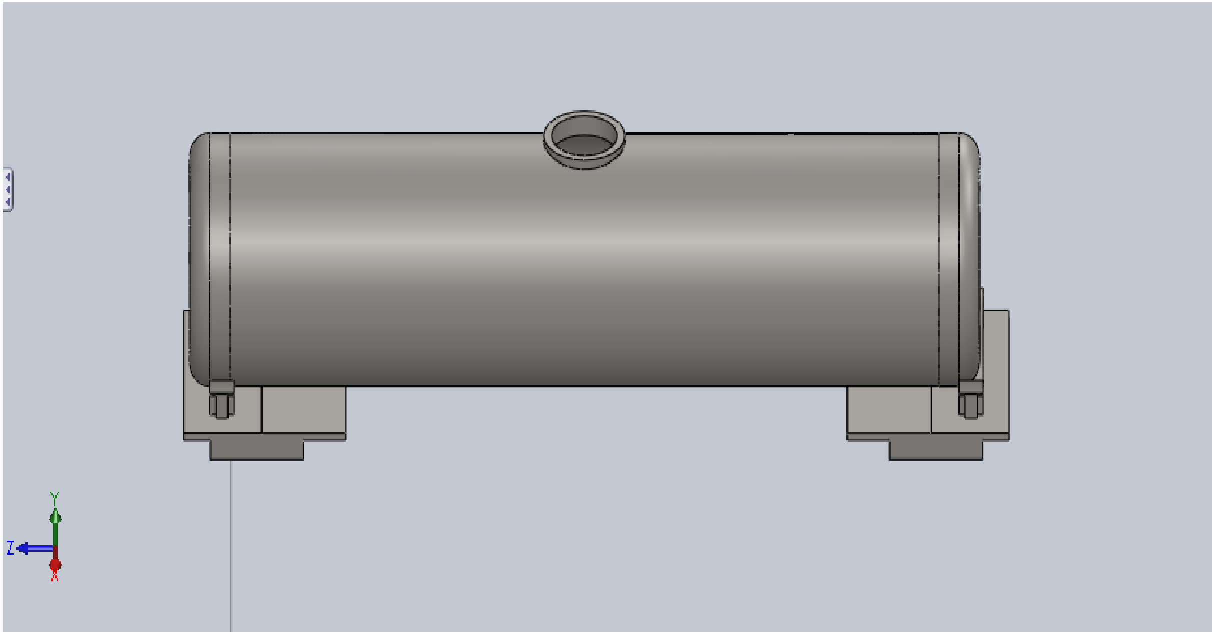


Figure 11: model cylindrical Tank by Solidworks

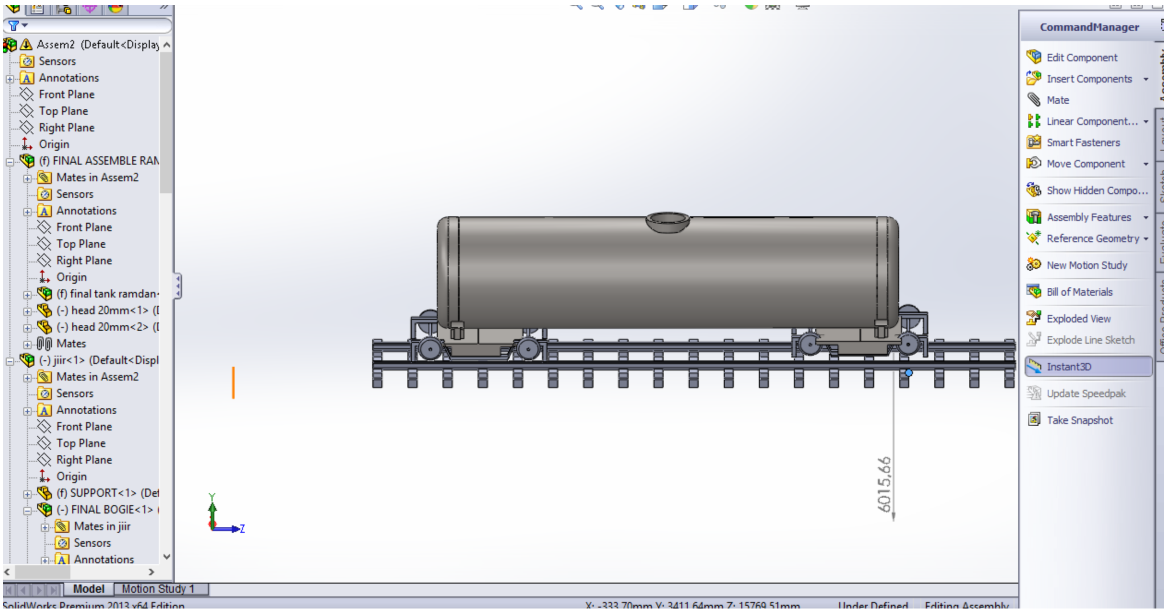


Figure 12: model cylindrical Tank in straight track by Solidworks.

4.2. Applying Load and Boundary Condition

After completion of the finite element model of the cylindrical oil tank wagon, it was necessary to apply load and boundary condition to the model. The analyzed railways tank wagon is subjected to the following load:

-The initial temperature of the tank wagon is 42℃

-The pressure applied to the tank is 1.5 MPas.

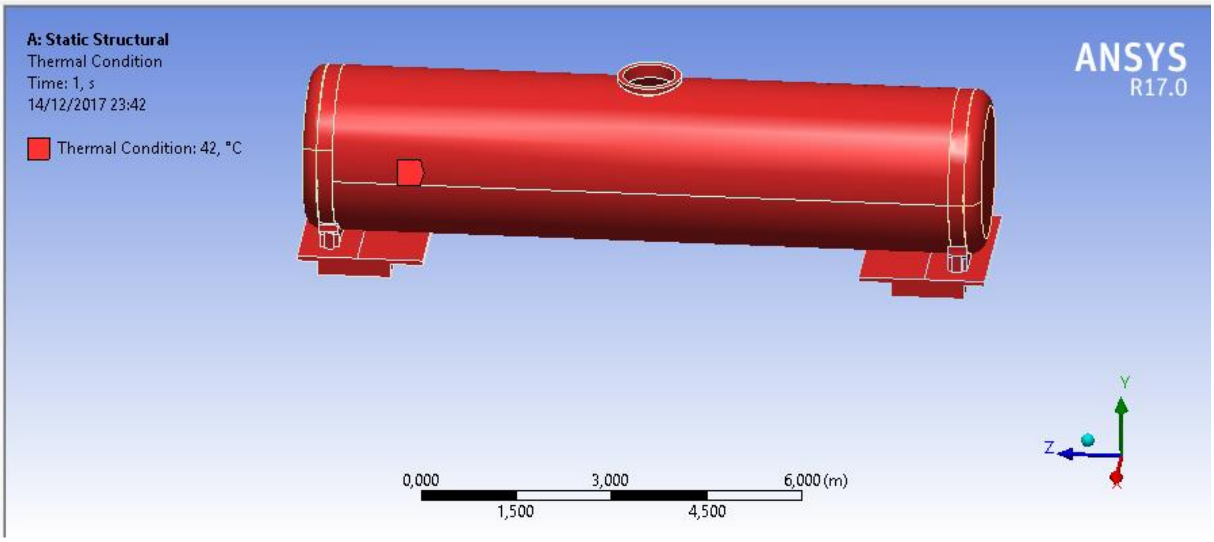


Figure 13: Temperature Applied

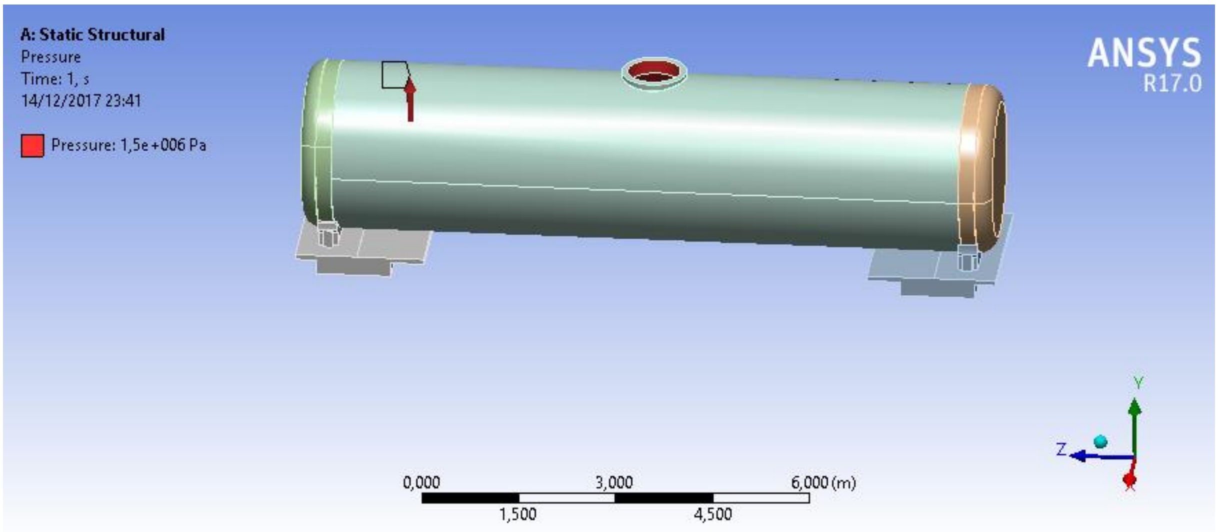
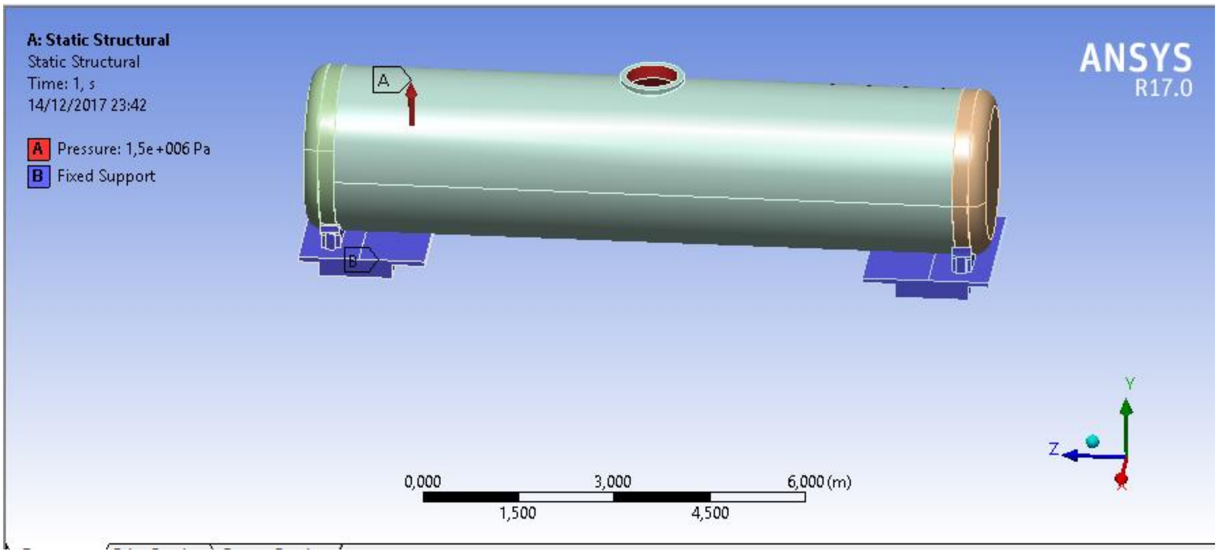


Figure 14: pressure Applied

4.3. Structural load and boundary condition

The static structural load and the boundary condition are the following :

-The fixied support is the fixed part.



.

Figure 15: Structural load and boundary condition.

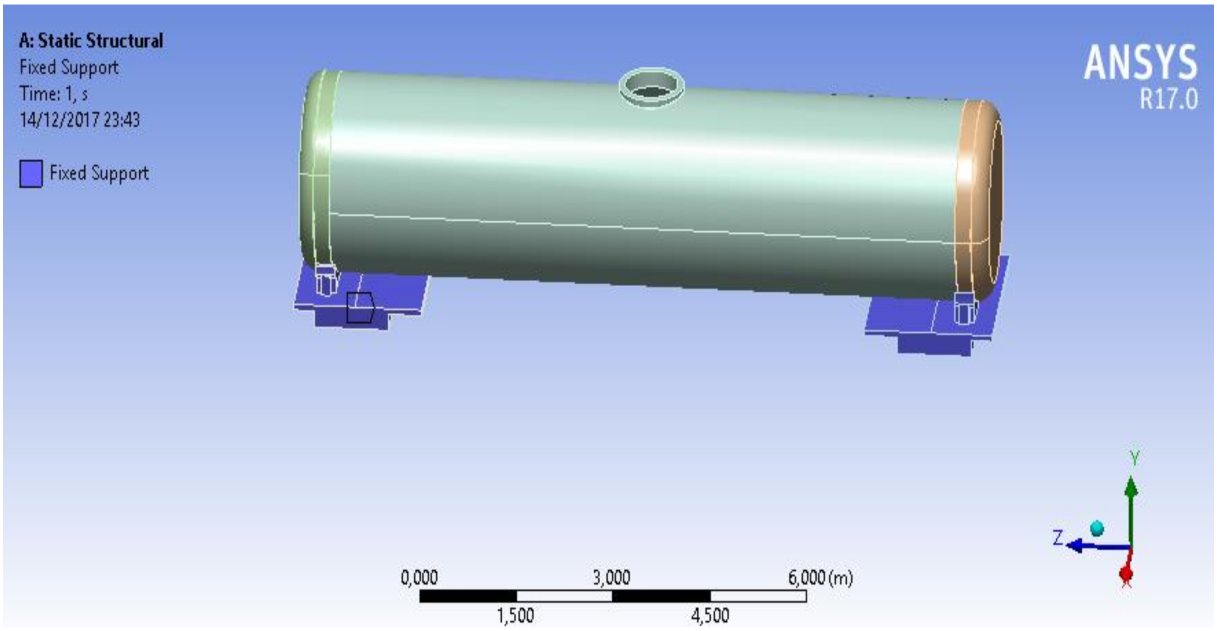


Figure 16:fixed support

4.4. Simulation

4.4.1. Mesh

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called element, which are connected together at a finite number of point called nodes. A network of these elements is known as a mesh. Mesh is discretizing the solid object to finest parts to perform the analysis to get the precise value at each and every element of the meshed object. The size of the meshing type is fine size mesh; the element is30718 and the Node is 61288.

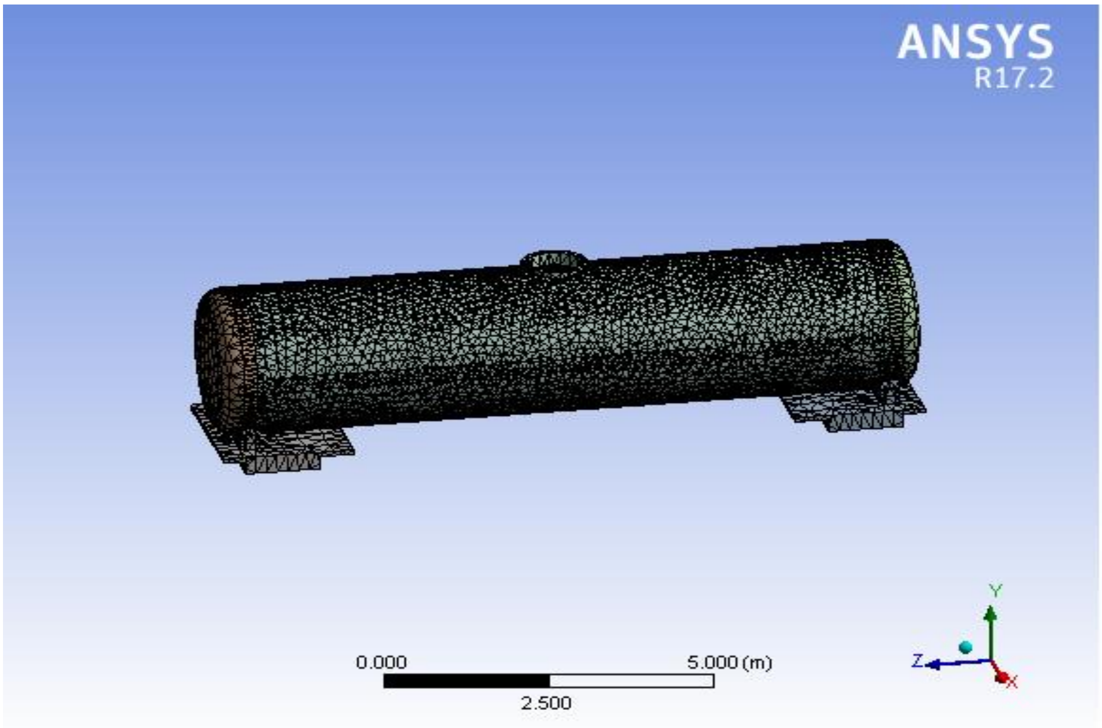


Figure 17: Fine sized fluent meshed Cylindrical tank

4.4.2. Static Analysis

A static structural analysis is determining the total deformation, equivalent (von misses) stress, Equivalent (von misses) strain, fatigue life, Damage, safety factor, etc….

4.4.2.1. Stress

In mechanics a cylinder stress is a stress distribution with rotational symmetry that is which remain unchanged of the stress object is rotated about some fixed. Stress is defined as average forces per unit area that some particles of a body exert on are adjacent particle, across an imaginary surface that separates them.

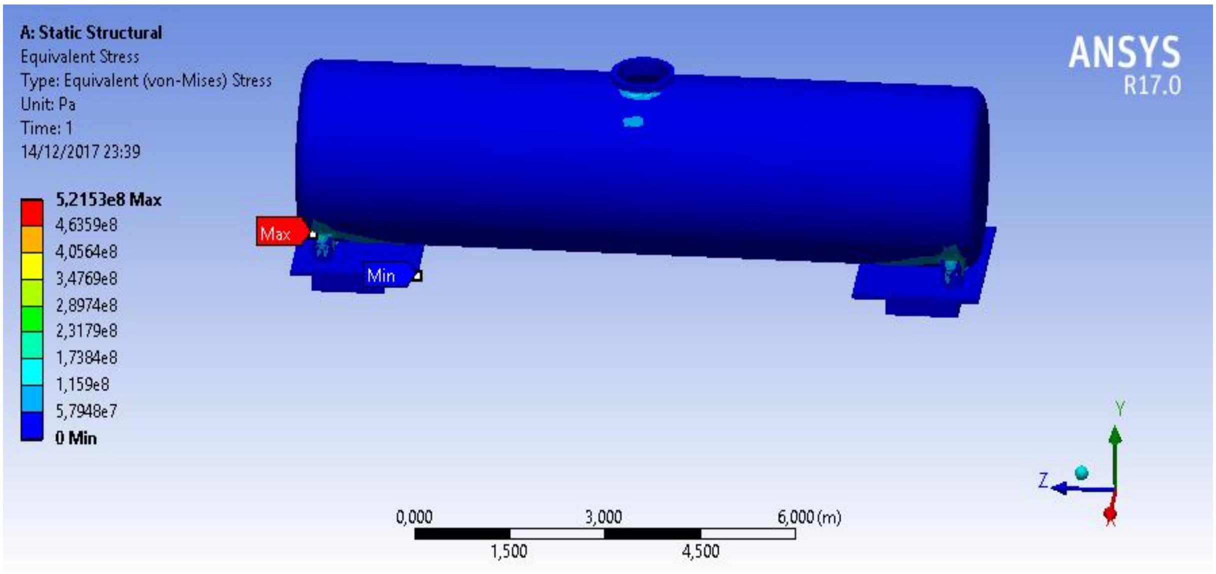


Figure 18: Equivalent (von-mises) stress.

As shown in above figure, Maximum Equivalent (von miss) stress is 521.53MPas Max , and the minimum Equivalent (von miss) stress is 57.949 MPas Min.

B. Strain

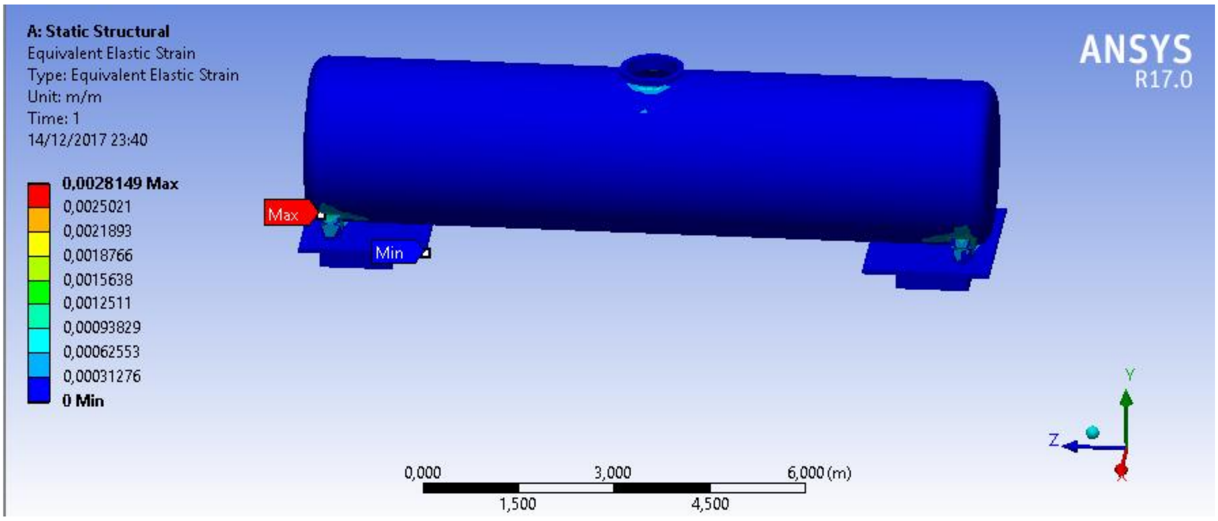


Figure 19: Equivalent Elastic Strain

As shown in above figure, the maximum Equivalent elastic strain is 0.0028149 and the minimum Equivalent elastic strain is 0.000312.

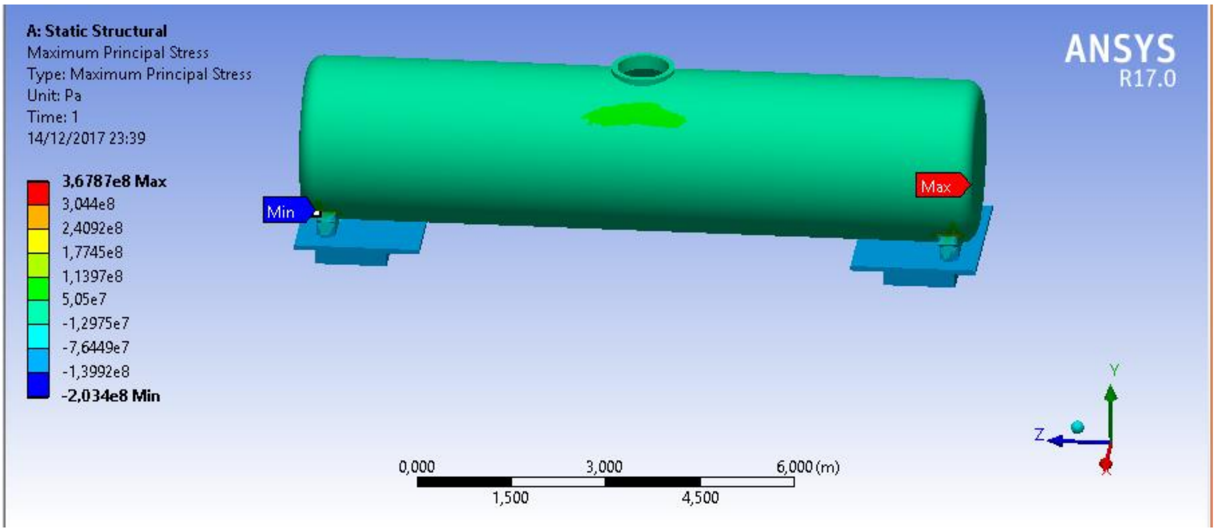


Figure 20: Maximum principal stress

As shown in above figure, the maximum principal stress is 367.87MPa and the minimum principal stress is -203.4MPa.

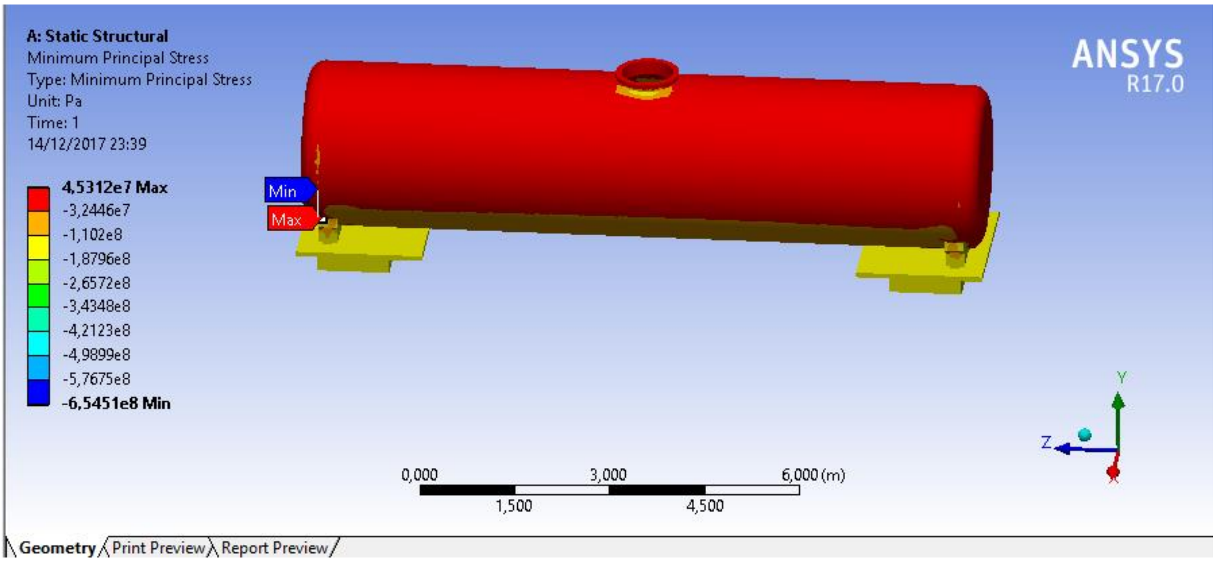


Figure 21: Minimum principal stress

As shown in above figure, the minimum principal stress is 45.32 MPaMAX, and -654.51MPa MIN.

4.5. Fatigue life Analysis

Fatigue Life can be over the whole model or scoped just like any other contour result in Workbench, This result contour plot shows the available life for the given fatigue analysis. If loading is of constant amplitude, this represents the number of cycles until the part will fail due to fatigue. If loading is non-constant, this represents the number of loading blocks until failure.

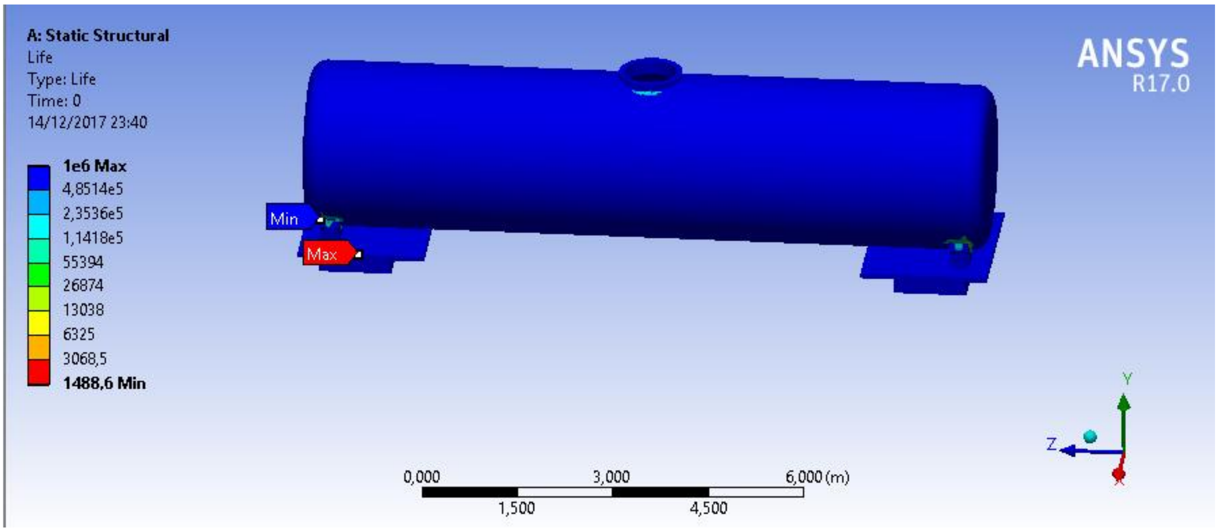


Figure 22: life cycle

As shown in above figure, the maximum life cycle is 1e6 Max , and the minimum life cycle is 1488.6Min.

Fatigue Safety Factor is a contour plot of the factor of safety with respect to a fatigue failure at a given design life. The maximum Factor of Safety displayed is 15. Like damage and life, this result may be scoped. For Fatigue Safety Factor, values less than one indicate failure before the design life is reached.

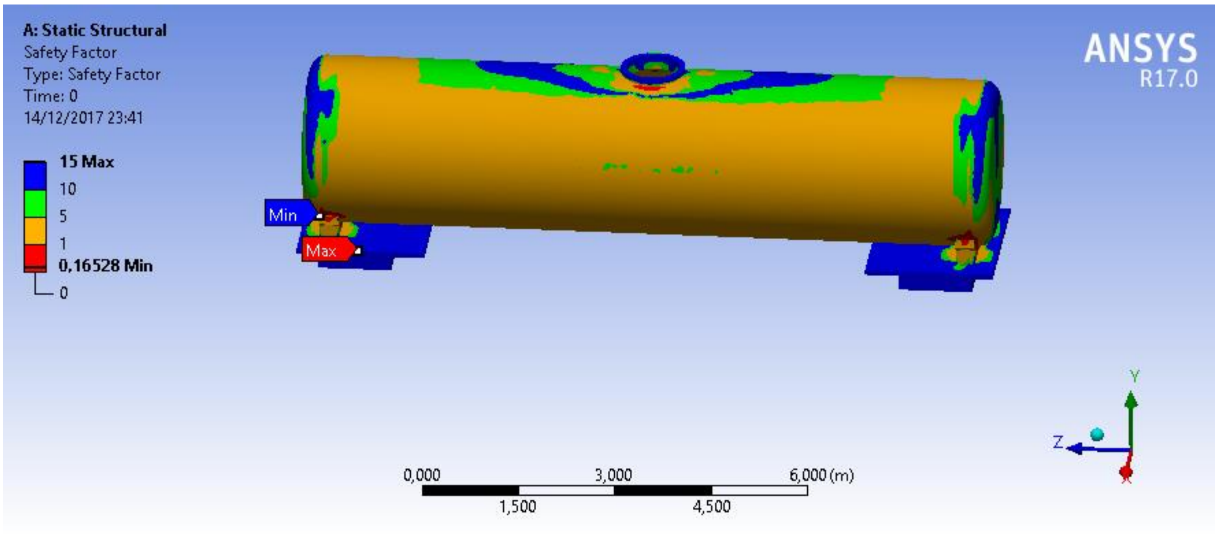


Figure 23: safety Factor

As shown in above figure, the maximum Safety Factor is 15 Max, and the minimum Safety Factor is 0.16528 Min.

Alternating stress:

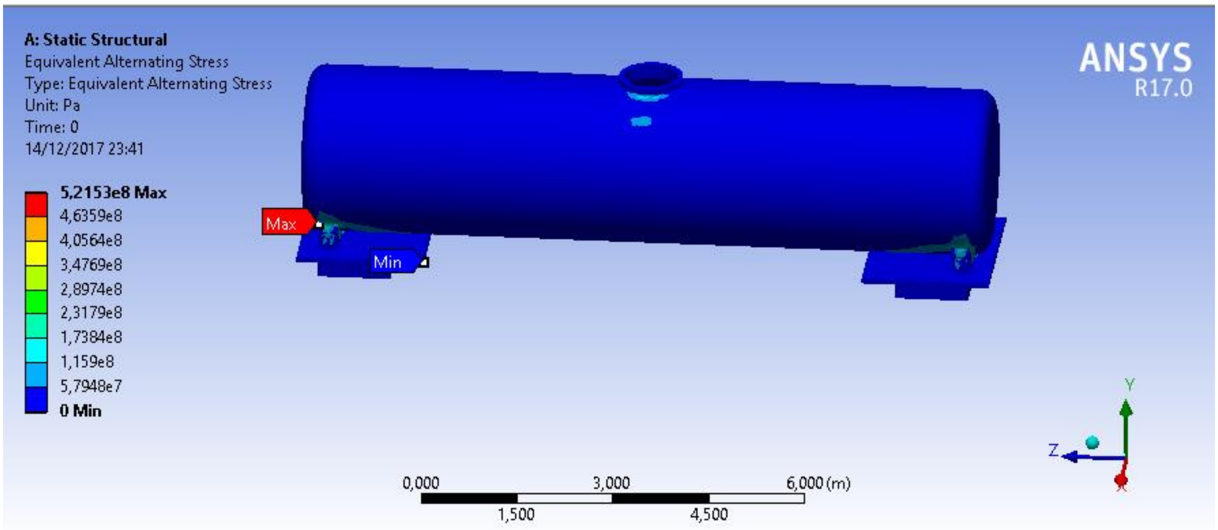


Figure 24: Alternating stress

As shown in above figure, Maximum alternating stress is 521.53MPa Max , and the minimum alternating stress is 57.94MPa Min .

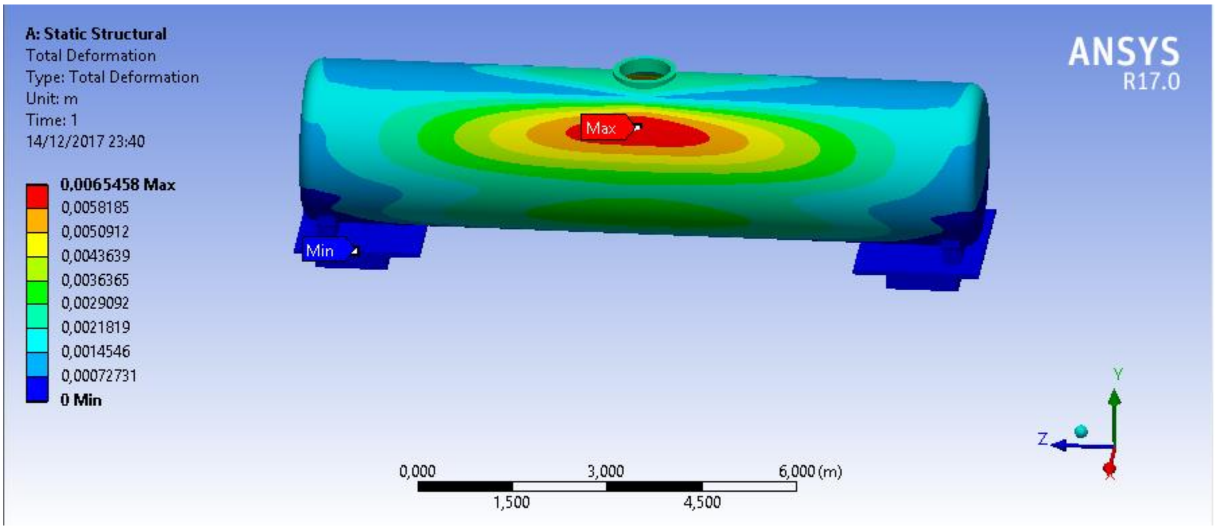


Figure 25: Total Deformation

As shown in above figure, Maximum Total Deformation is 0.0065458 Max, and the minimum Total Deformation is 0.00072731 Min.

Fatigue Sensitivity shows how the fatigue results change as a function of the loading at the critical location on the model. This result may be scoped. Sensitivity may be found for life, damage, or factor of safety. The user may set the number of fill points as well as the load variation limits.

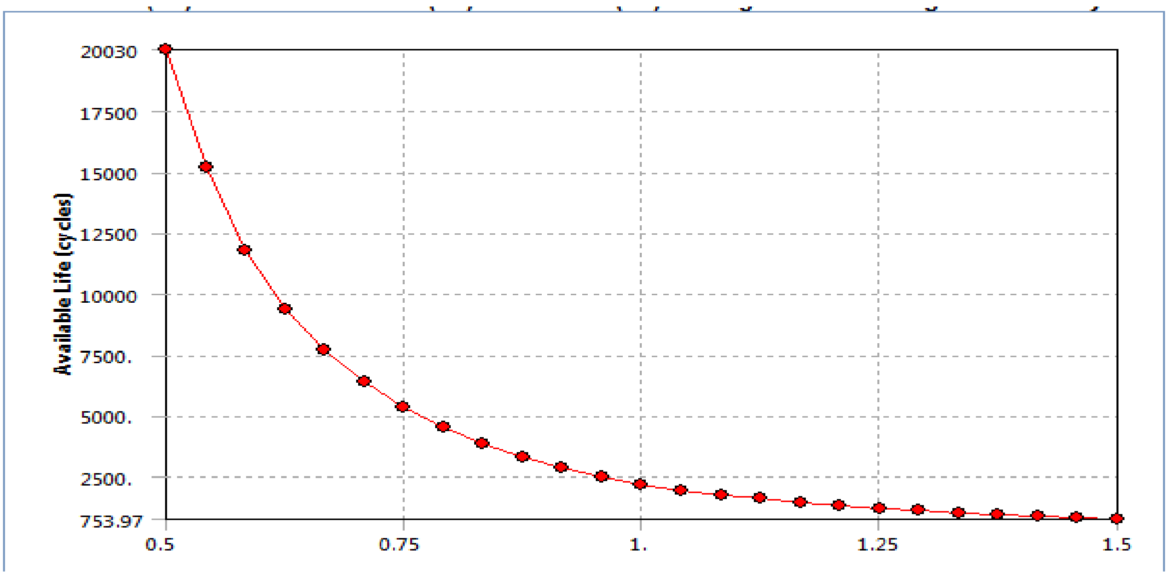


Figure 26: fatigue sensitivity

CHAPTER: FIVE

5. Result and Discussion

5.1. Result

The analysis was carried out by using finite element model consist of stress analysis, to determine the deformation of the cylindrical tank wagon; Modelling of the cylindrical tank was undertaken as illustrated in chapter Four. A detailed rail Tank wagon structure is idealized using the finite element analysis softwares ANSYS (Version17.2) and FEMFAT (version 5). Fine meshes are used to prevent any possible stress concentration effects.After preparing the model, meshing of the elements was undertaken and response spectrum analysis was conducted. The material properties used for the system are as shown also in Chapter Three. The temperature using to analyse the tank is ambient temperature of 42 ^{0}C, the pressure inside the tank is 1.5MPas.

5.1.1. Analysis forthe Cylindrical tank Material by ANSYS

Case 1: The currently used Tank Wagon material(from ERC)

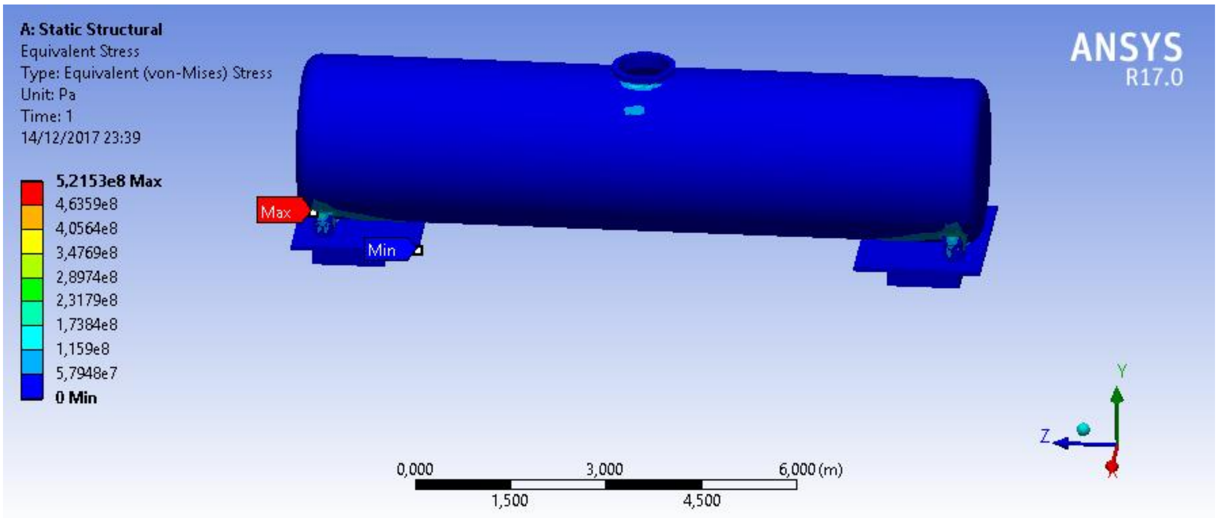


Figure 27: Equivalent (von miss) stress of Carbon steel

As shown in above figure, Maximum Equivalent (von miss) stress is 521.53MPas Max , and the minimum Equivalent (von miss) stress is 51.153MPas Min.

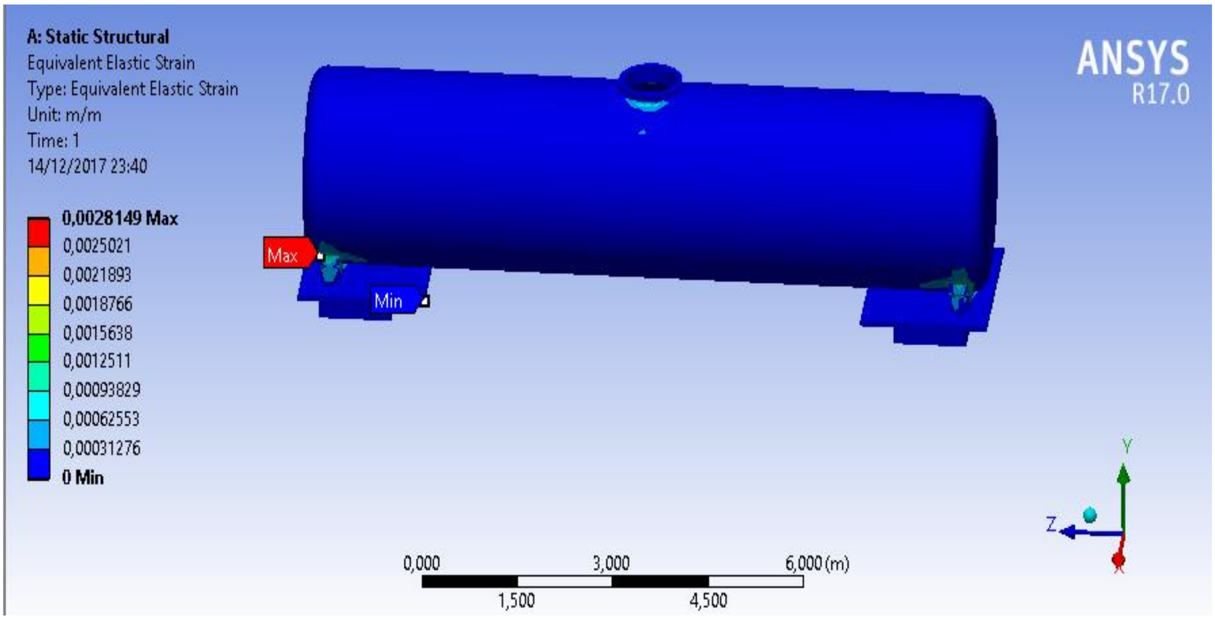


Figure 28: Equivalent Elastic Strain

As shown in above figure, the maximum Equivalent elastic strain is 0.0028149 and the minimum Equivalent elastic strain is 0.000312.

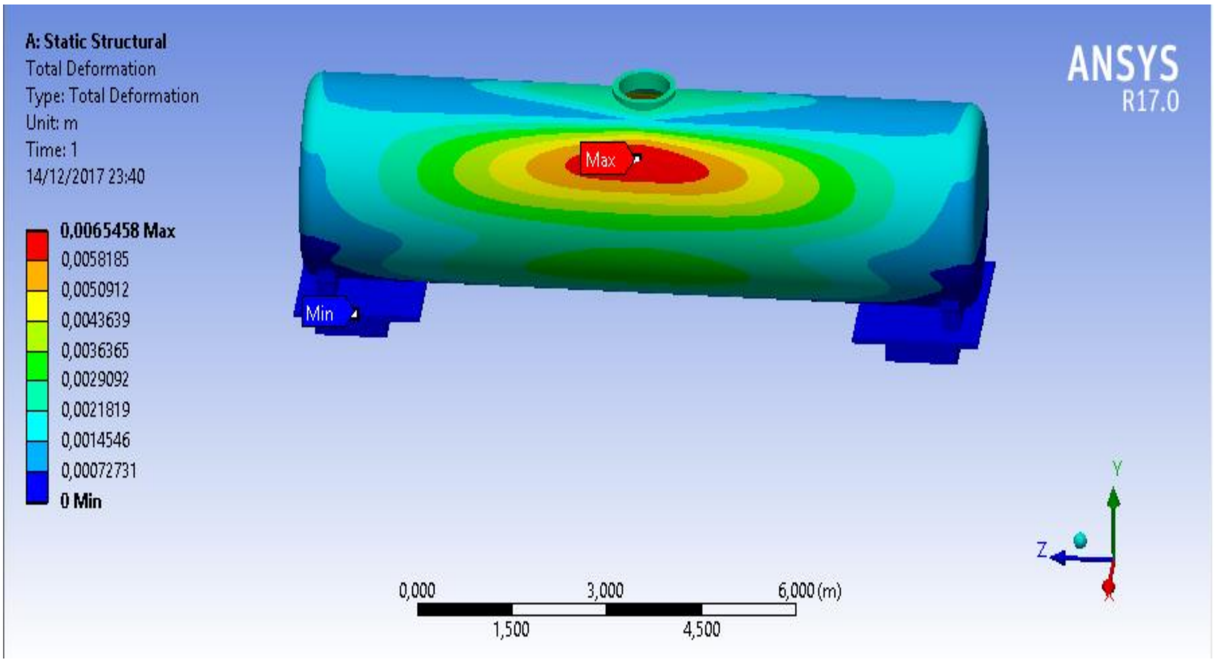


Figure 29: Total Deformation of carbon steel

As shown in above figure, Maximum Total Deformation is 0.0065458 Max, and the minimum Total Deformation is 0.00072731 Min.

Case 2: Deformation Comparing With AISI A656 High strength alloy steel

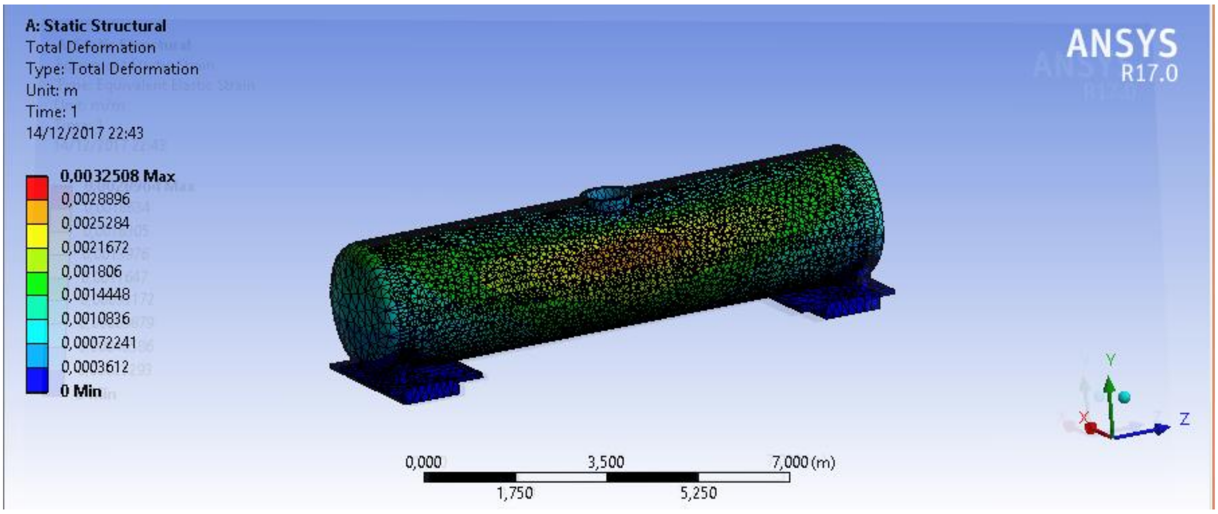


Figure 30: Total deformation of AISI A656 High strength alloy steel

As shown in above figure, Maximum Total Deformation is 0.0032588 Max, and the minimum Total Deformation is 0.0003612Min .

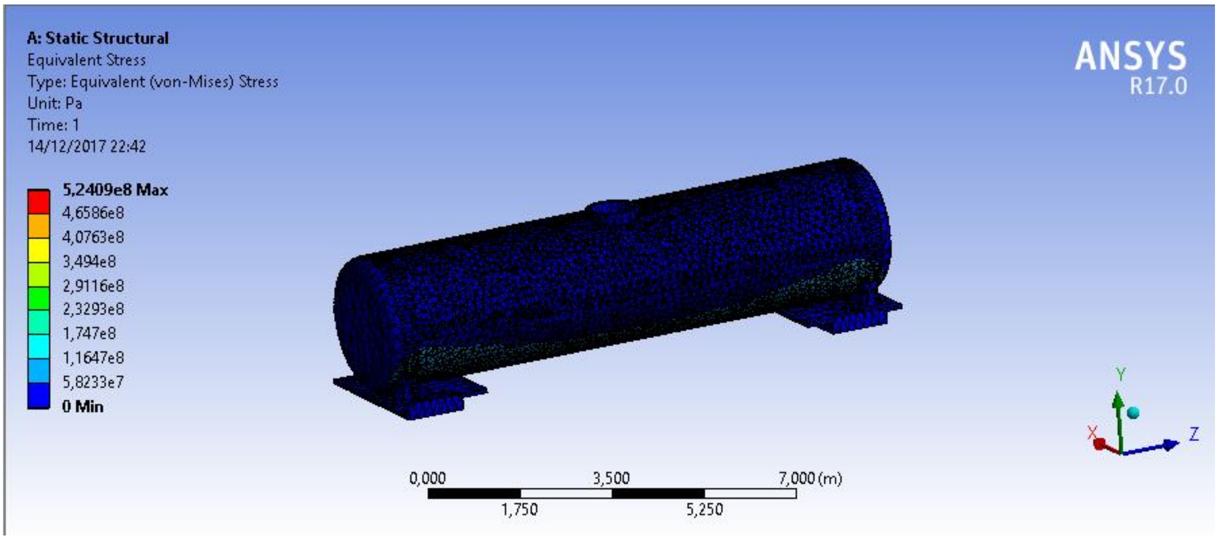


Figure 31: Equivalent (von miss) Stress of AISI A656 High strength alloy steel

As shown in above figure, Maximum Equivalent (von miss) stress is 524.09MPa Max , and the minimum Equivalent (von miss) stress is 58.23MPa Min.

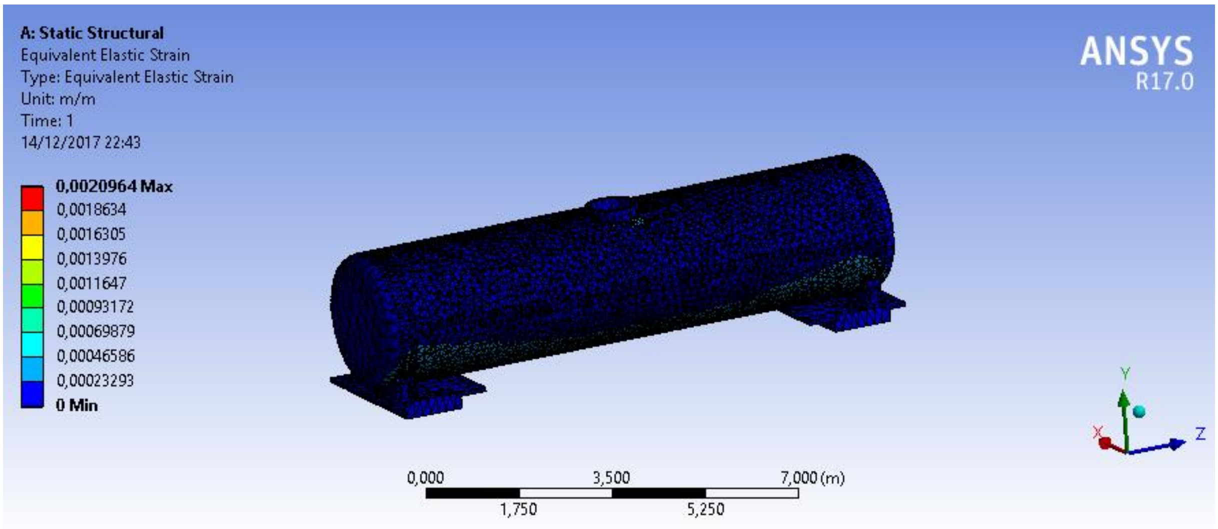


Figure 32: Equivalent (von miss) Strain of AISI A656 High strength alloy steel

As shown in above figure, the maximum equivalent elastic strain is 0.002096 Max and the minimum equivalent elastic strain is 0.0002329 Min.

Table 5: Maximum deformation comparing with the materials

No	Material		Total deformation (m)	Stress (MPas)	Strain(MPas)
1	Carbon steel	Max	0.0065458	521.53	0.0028149
		Min	0.0007273.	51.153	0.000312
2	AISI	Max	0.0032588	524.09	0.002096
		Min	0.0003612	58.23	0.0002329

5.1.2 FEMFAT RESULT

Case 1: Fatigue analysis by currently used material (ERC)

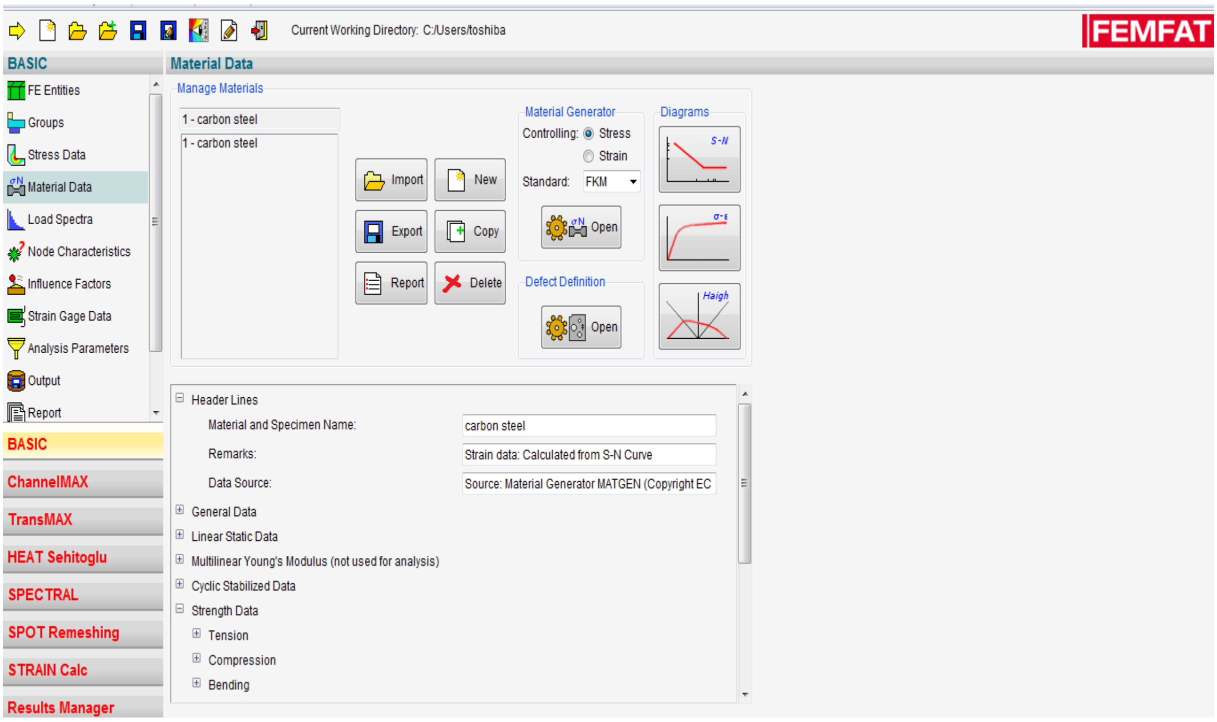


Figure 33: imputing material data in FEMFAT FILE

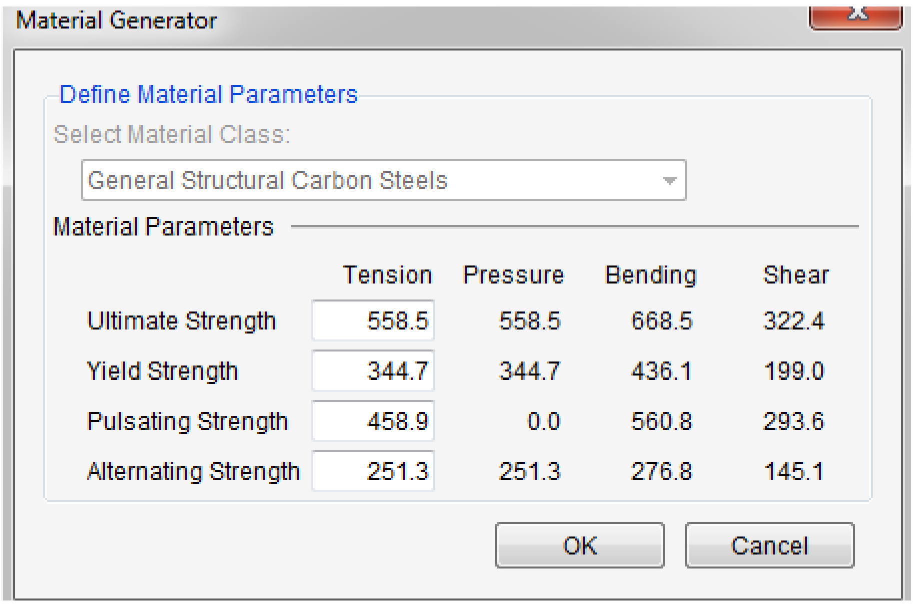


Figure 34: material parameters

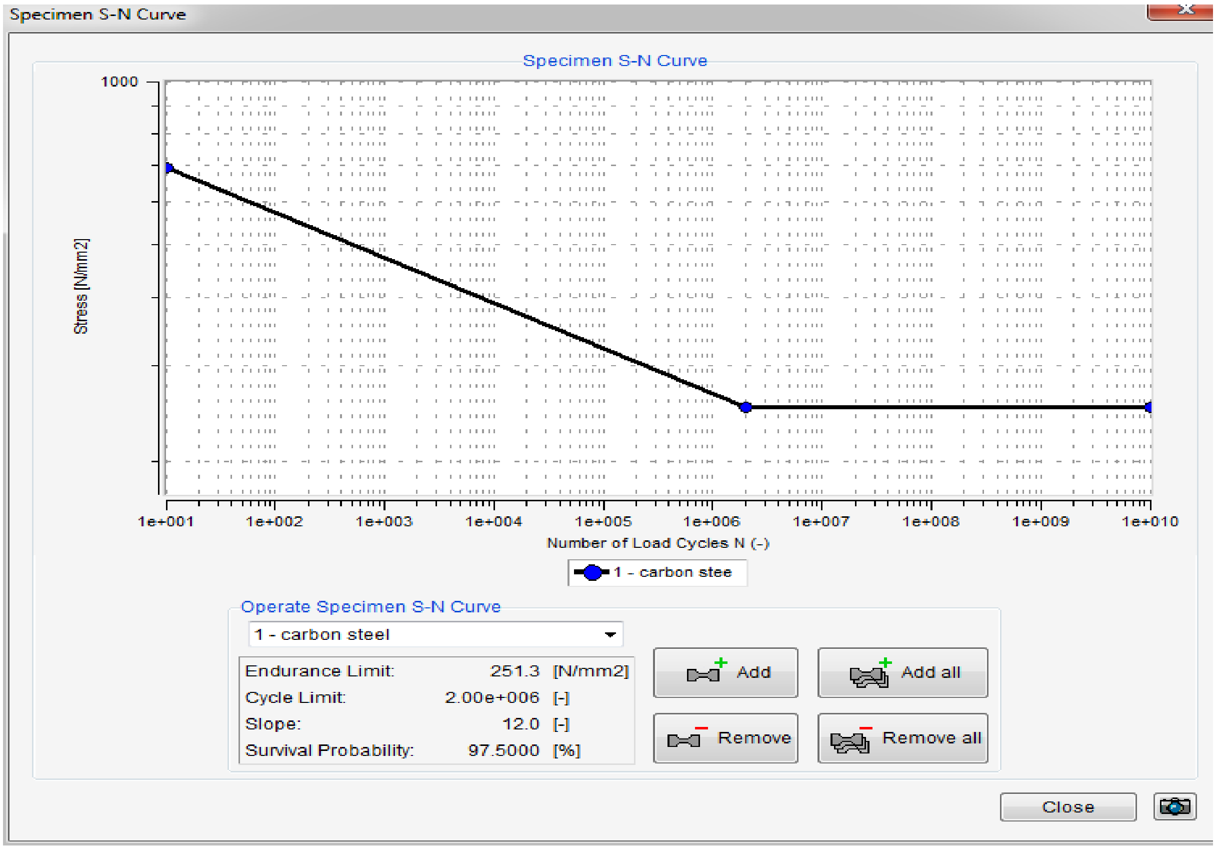


Figure 35: Specimen S-N curve

Case 2: Fatigue analysis by selected material AISI A656 high strength alloy steel

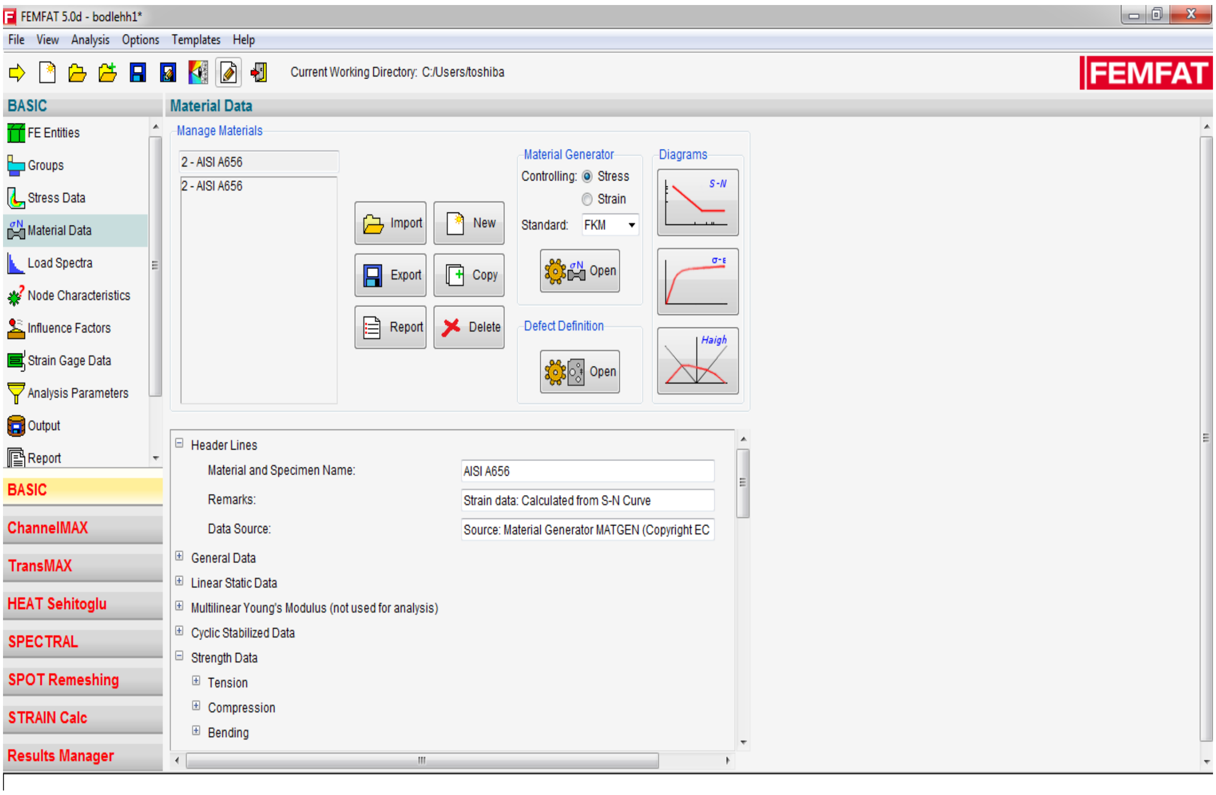


Figure 36: manage materials from selecting material

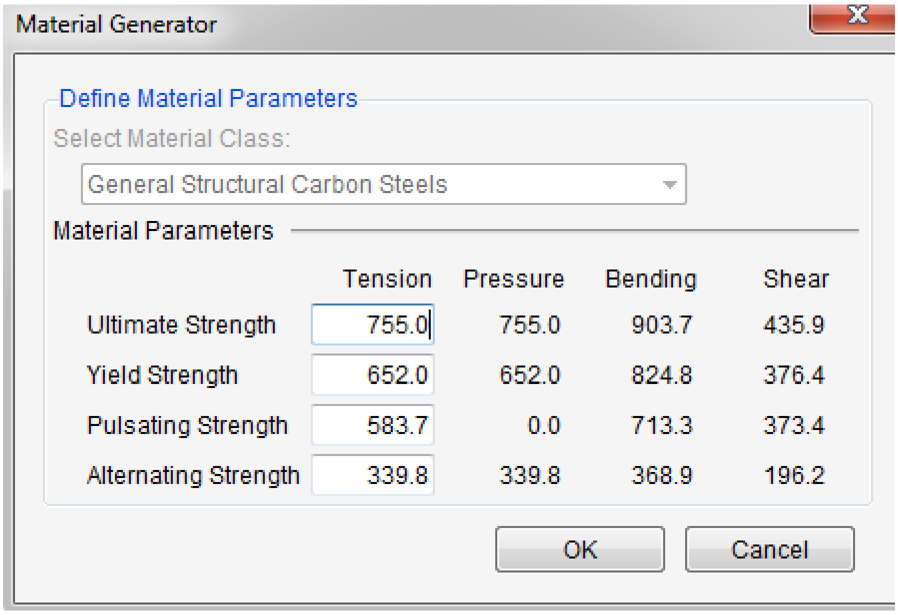


Figure 37: define material parameters

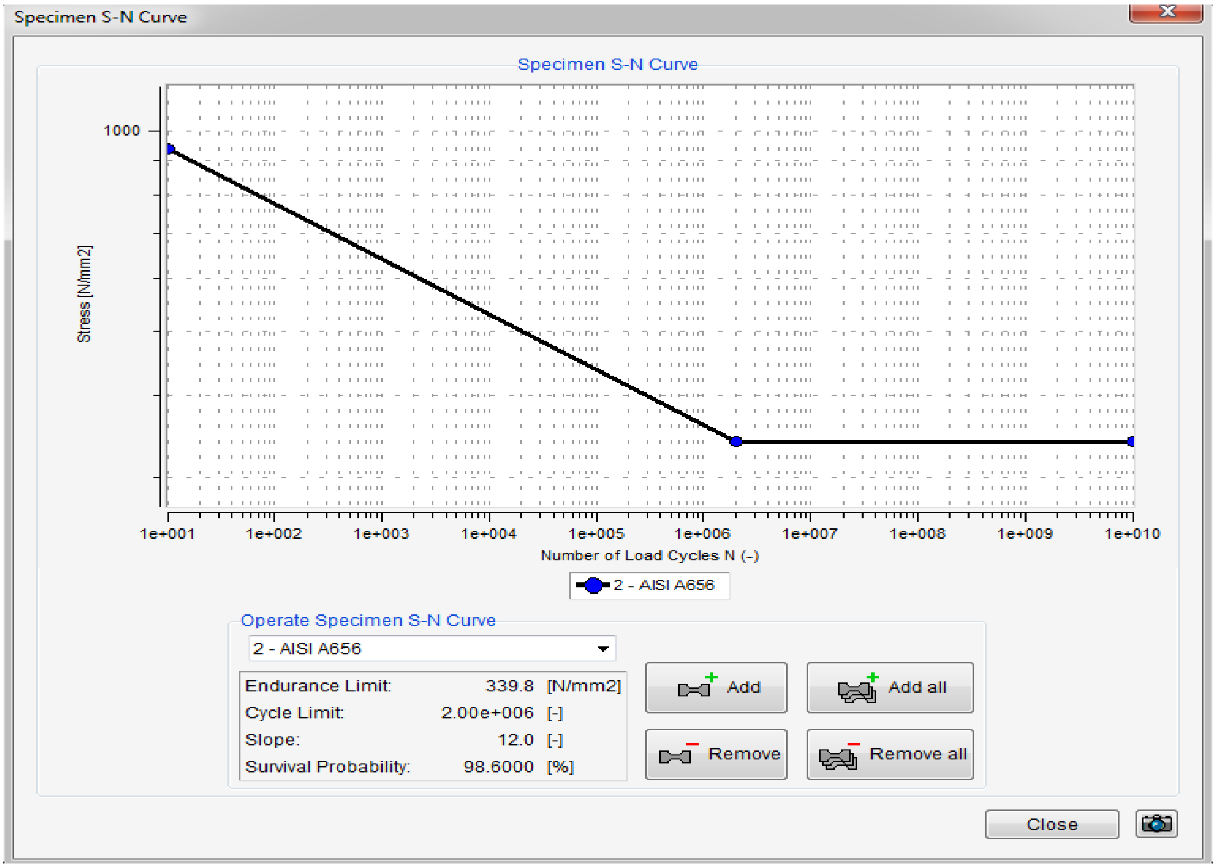


Figure 38: specimen S.N curve from selected material

Table 6: FEMFAT result analysis

No	Material	Endurance limit (N/mm2)	Survival propability (%)
1	Carbon steel	251.3	97.5%
2	AISI	333.8	98.6 %

5.2. Discussion

The above result of the analysis performed of cylindrical oil tank wagon using the ANSYS software (version 17.2) and FEMFAT version 5.0 with the materials analysis are show the Table below.

The case of stress,deformation Analysis by ANSYS software

In the analysis cylindrical tank wagon has been used in different material byanalysis ANSYS software, this section of the paper specifies the result obtained from the ANSYS software based on the maximum deformation. In this case the maximum deformations are0.0065458 Max and 0.0032588 Max. So the smaller deformation is AISI A656 High Strength Alloy Steel with 0.0032588 Max maximum deformations. In comparision the deformation with the two material, we get the carbon steel materiel is bigger than the AISI A656 High strength alloy steel materiel.

The case of fatigue Analysis by FEMFAT software

The fatigue analysis of the currently used material and the new materials, S.N curve is utilized. FEMFAT (finite element fatigue) software above compare the existing material with develop one by showing different endurance limit and survival probabilities. The conventional material is below than the new material, therefore that developed material will be more difficult to break and stronger than conventional one.

CHAPTER: SIX

6. CONCLUSION, RECOMMENDATION, AND FUTURE WORK

6.1. Conclusion

In this study, the responses of the stress and fatigue analysis of cylindrical tank wagon are determined under static, and the results are assumed to be significant. The analysis can include stress, strain and fatigue. The stress and deformation are reduced for the new material.

Using the ANSYS finite element software, elements depicting the properties of the cylindrical tank wagon were selected. From the results obtained using the finite element method and comparing those with changing the materials the following conclusions can be drawn.

The stress and deformation of cylindrical oil tank wagon value is reduced for the new material. From the results obtained in the stress and the deformation analysis, the two cases have different total deformation. The total deformation distribution of ERC tank is high in comparison to new material of the tank analysis.

The conclusions drawn from this research are:

- The analysis of stress, deformation analyisis of the cylindrical oil tank wagon in ANSYS software where the maximum deformation of the currently used material is0.0065458 Max andthe AISI A656 high strength alloy steel is 0.0032588 Max, so the smaller total deformation all of them is AISI A656 High strength alloy steel with the maximum deformation is 0.0032588 Max.
- . FEMFAT reveal that the selected material has higher endurance limit and survival probabilities than current material.
- Finally by analyze the material of the cylindrical tank, the result has great importance to minimize the total deformation of the tank by changing the materials to AISI A656 High Strength Alloy Steel.

6.2. Recommendation

This researcher recommends that the cylindrical tank wagon be designed as stiff as possible to resist the stress and deformation. The appropriate material must be selected for the tank to reduce the stress and deformation of the cylindrical tank wagon. However, this tank, which is wagon structure, is made of carbon steel of different profile, is believed to be available in the market with low cost. Even thought, we have already analysedthe stress deformation of the tank wagon and obtained the result that the total deformation of AISI A656 High Strength Alloy Steel materials is smaller than the currently used materials analysis with ANSYS software, it is recommended to change the material.

6.3. Future works

In this thesis, the stress and deformation caused by cylindrical tank wagon are studied, intending to analyse thecurrently used material for ERC and AISI A656 high strength alloy steel material by ANSYS software. The thesis focuses on finite element analysis, but to improve the problem related to the deformation. The following future works are suggested to students of mechanical railway students who are interested in Freight Tank Wagon:

- Design and stress analysis of spherical tank.
- Stress and fatigue analysis of cylindrical tank wagon with hemispherical head.

7. APPENDIX

7.1 : Constant Amplitude Load

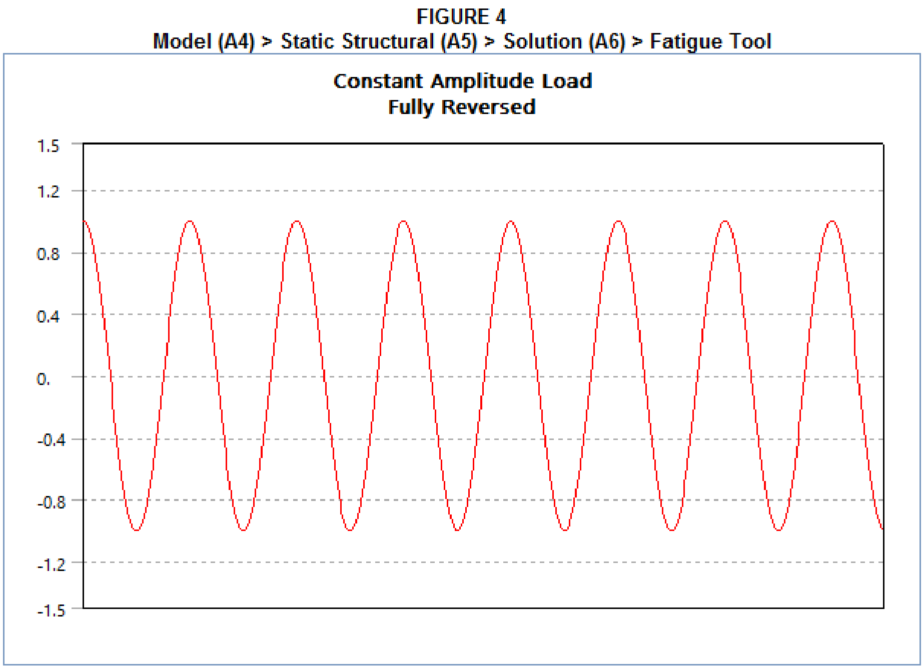


Figure 7.1: Constant Amplitude Load

7.2 : Mean Stress Correction Theory

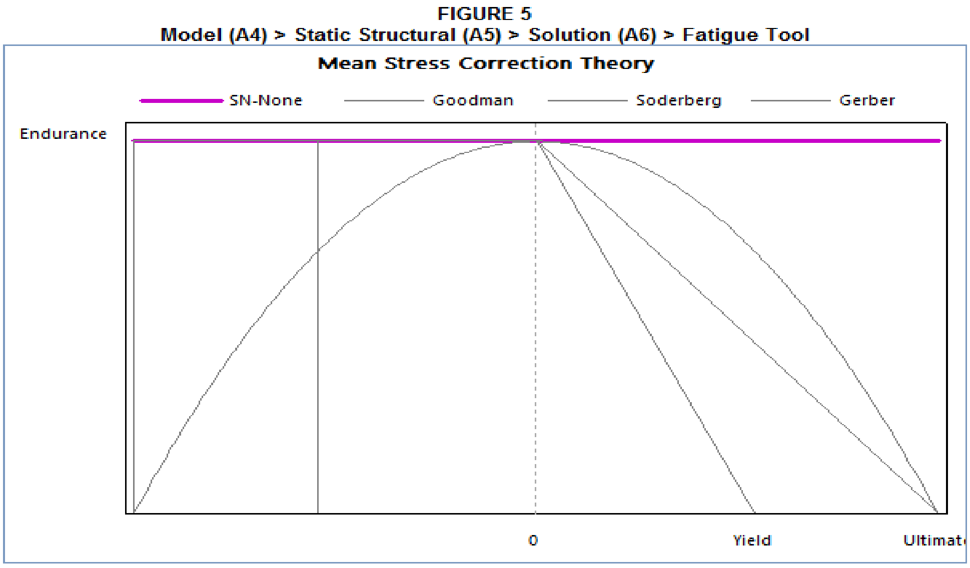


Figure 7.2: Mean Stress Correction Theory

7.3. Pressure

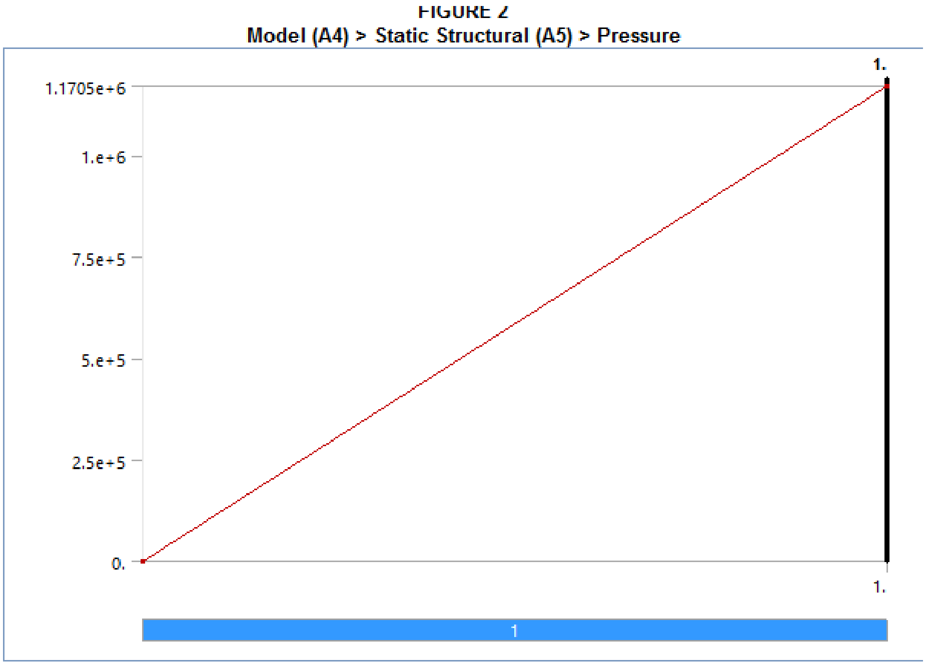


Figure 7.3: pressure

7.4. Thermal Condition

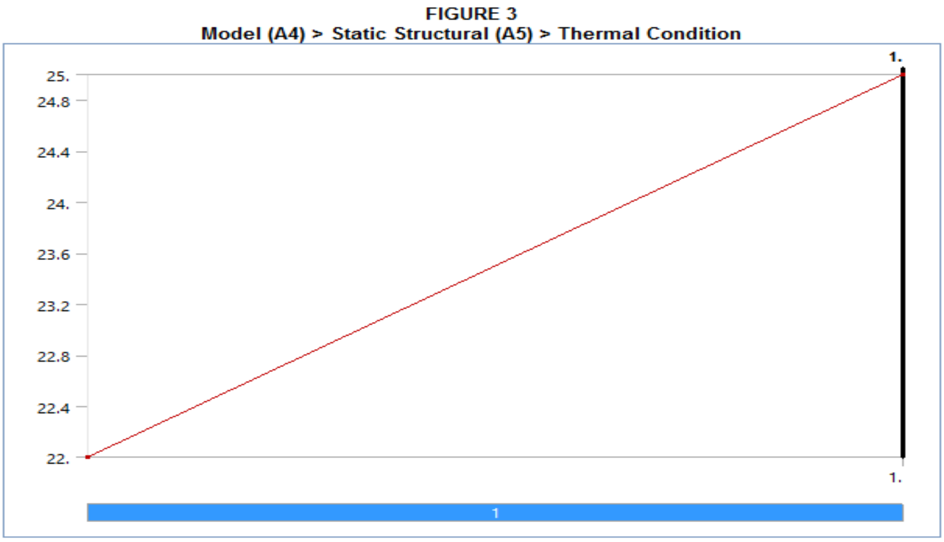


Figure 7.4: Thermal condtion

8. Reference

[1]. Early Oil Transportation: A Brief History.

[2]. A brief history of tank container.

[3]. International Conference on "Design, inspection, maintenance and operation of cylindrical steel tank and pipelines"Prague, Czech republic - October 8-10, 2003.

[4]. Influence of Barrel Damages on life time of tank wagon " 8th International DAAAM Baltic Conference "Industrial Enfineering 19-21 April 2012 ,Tallin ,Estonia.

[5]. A. Flores-Herrera, J. M. Sandoval-Pineda " Dynamic Analysis of Railroad Tank Car Under motion scenario Yaw and Sway " SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 1 Issue 7 Nov 2014 ISSN: 2348 – 8360 www.internationaljournalssrg.org Page 28.

[6]. Y.Y. Semin, M.N. Kulyapin (CJSC "VNIIST-Diagnostika", RUSSIA)" STUDY OF STRESS-DEFORMED STATE IN DENTS AREA ON THE WALL OF VERTICAL STEEL TANKS ".

[7]. Margit Weltschev, Stephanie Schwarzer, Frank Otremba"Comparison of the Operating Life of Tank Containers, Tank Vehicles and Rail Tank Cars for the Carriage of Dangerous Goods in Practice, Analysis of Causes of Damage".

[8]. V.N. Skopinskyand A.B. Smetankin, "Modelling and Stress analysis of nozzle connections in Ellipsoidal heads of Pressure vessels under External loading" International Journal of Applied Mechanics and Engineering, 2006, vol.11, No.4, Page no. 965-979.

[9]. Prof. Vishal V. Saidpatil,Prof. Arun S. Thakare ,"Design & Weight Optimization of Pressure Vessel Due to Thickness Using Finite Element Analysis"International Journal of Emerging Engineering Research and Technology Volume 2, Issue 3, June 2014, PP 1-8 ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online).

[10]. Ahmed Ibrahim, YeongRyu, Mir Saidpour 'Stress Analysis of Thin-Walled Pressure.

[11]. Vessels' Modern Mechanical Engineering, 2015, 5, 1-9Published OnlineFebruary2015 in SciRes.http://www.scirp.org/journal/mmehttp://dx.doi.org/10.4236/mme.2015.51001.

[12]. Apurva R. Pendbhaje,and Mahesh Gaikwad "DESIGN AND ANALYSIS of PRESSURE VESSEL "International Journal of Innovative Research in Technology & Science(IJIRTS) ISSN:2321-1156.

[13]. V. V. Wadkar, S.S. Malgave, D.D. Patil , H.S. Bhore , P. P. Gavade"DESIGN AND ANALYSIS OF PRESSURE VESSEL USING ANSYS "Journal of Mechanical Engineering and Technology (JMET) Volume 3, Issue 2, July-Dec 2015, pp. 01-13, Article ID: JMET_03_02_001 Available online .

[14]. R.C. Carbonari, P.A. Muñoz-Rojas, E.Q. Andrade, G.H. Paulino, K. Nishimoto, E.C.N. Silva, “Design of pressure vessels using shape optimization: An integrated approach”, International Journal of Pressure Vessels and Piping, 88 (2011), pp. 198-212.

[15]. PuneetDeolia, Prof. F.A. Shaikh. ‘Burst Pressure Analysis of a Pressure Vessel’ International journal of research in advent technology.

[16]. Zainul Huda, Muhammad Hani Ajani ; ‘Evaluation of Longitudinal and Hoop Stresses and a Critical Study of Factor of Safety (FoS) in Design of a Glass-Fiber Pressure Vessel ‘ World Academy of Science, Engineering and Technology International Journal of Civil, Environmental, Structural, Construction and Architectural Engineering Vol:9, No:1, 2015

[17]. International.National Transportation Safety Board document (2016), "DOT-111 Tank Car Design" (http:/ / www. ntsb. gov/ news/events.

[18]. Farhadnabhani ,temilade ladokun ,and vahidaskari ; ‘Reduction Of Stress in Cylindrical Pressure Vessel Using Finite Element Analysis’’. school of sccience and engineering ; middlisbring TS13BA,UK.

[19]. SourabhLawate, B. B. Deshmukh ; ‘Analysis of Heads of Pressure Vessel ’. International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 4, Issue 2, February 2015.

[20]. KirtikumarTamboli , 'Fatigue Analysis of Pressure Vessel by FEA Techniques'.International Journal of Engineering Trends and Technology (IJETT) – Volume 13 Number 1 – Jul 2014 ISSN: 2231.

[21]. PrashantNanavareandAbhayUtpat, 'Fatigue Analysis of 6300 Liters Pressure Vessel By Using Cyclic Service' International Journal of Modern Trends in Engineering and Research ISSN No.:2349-9745, Date: 2-4 July, 2015.

[22]. GW2 Tank Wagon operation instruction data from ERC.

[23]. $ylvesterSuda and Bernar**d** Keisch ; 'Temperature**Ef**fects**and** Corrections in Volume MeasurementsBased on Liquid-*L*evelDetection 1'.