Covenant Journal of Engineering Technology (CJET) Vol. 1, No. 2, Sept. 2018

An Open Access Journal Available Online

Finite Element Analysis of Displacement and Von-Mises Stress in Cylindrical Liquified Petroleum Gas Pressure Tank

C.U. Ugochukwu[1*], O.O. Oluwole[1], K.M. Odunfa[1]

[1]Department of Mechanical Engineering,
University of Ibadan, Ibadan Nigeria.

Abstract**:** Increase in demand of liquefied petroleum gas (LPG) has led to development of LPG facilities throughout the world. The limitation of ASME standard in the design of pressure vessels and reoccurring cases of gas plant, gas cylinder explosions led to this research. In this research, finite element method was used to investigate the displacements, deflections and Von-Mises stresses in a cylindrical liquefied petroleum gas pressure tank with respect to plate thickness at different operating pressures and ambient conditions. A cylindrical pressure tank made of ASTM A516 Grade 70 with thickness; 2mm, 5mm, 10mm, 20mm and 30mm was selected for the analysis with plain strain condition assumptions. ANSYS was used to generate the mesh model of the liquefied petroleum gas pressure tank and conduct the finite element analysis. The displacement, deflection and Von-Mises stress showed an inverse relationship with the tank section shell thickness while varying the LPG pressure; 0.5MPa at 20^0C, 0.91MPa at 40^0C and 1.55MPa at 60^0C respectively. It was also observed that the factor of safety showed a linear relationship with increasing shell thickness. For each operating pressure, a minimum shell thickness was deduced. This minimum thickness was at a Von-Mises stress which falls below the materials yield stress and allowable stress. Therefore, the vessel will not fail once operated at or above the minimum pressure tank shell thickness. The effect of weldment along the seams of vessel was not carried out in this research work. Sharp edges are stress raisers, also there is possibility of stress been developed at the inlet and exhaust valves of the pressure tank. The effect of stress at this points on the vessel were not considered for this research work.

Key-words— LPG, ANSYS, Finite Element Method, Von-Mises Stress.

1. Introduction

Liquefied petroleum gas (LPG) is a derivative of two large energy industries: natural gas processing and crude oil refining. Worldwide, natural gas processing is a source of approximately 60%, while crude oil refining contributes 40% of LPG produced (Foramfera, 2016). The main components of liquefied petroleum gas are propane and butane. LPG is colourless and odourless, but commercially odorized with ethyl mercaptan so that it can be detected when it has reached one-fifth the concentration needed for an explosion [2].

The Nigeria LNG Limited has reserved 250,000 metric tonnes per annum for the domestic market with a projection of 3 million metric tonnes per annum within five years [3]. Due to the growing demand for LPG, companies are rapidly developing facilities across the LPG value chain.

Liquefied Petroleum Gas is stored in pressure vessels. These containers are either cylindrical or spherical. While cylindrical vessel has ease of manufacture, spherical vessel has distinct advantage of less floor area coverage and high-pressure capability [4]. Despite these advantages of spherical vessels, the complexity of design limits their effective utilization. As the size of spherical vessels increases, high pressure is developed towards the base of the sphere. Hence, LPG is often stored, transported and distributed in cylindrical pressure vessels. The head of the vessel is of various kind of configuration which includes; flanged, torisherical, ellipsoidal and hemispherical [5]. When a pressure vessel is under load, stress is developed on the walls of the container. A number of stress theories, also called “yield criteria,” are available for describing the effects of combined stresses [6]. A material will yield or fails when it Von-mises stress is at a critical value which is known as the yield strength. The yield criterion is compared with experimental values to know if failure will occur.

The American Society of Mechanical Engineers (ASME) provides codes and simple formulas that regulate the design and construction of pressure vessels [7]. ASME standard is a generalization of simple formulas and has limitation in terms of specifying the actual fluid content on the pressure vessel. It does not put into consideration several actions or combination of actions such as local loads, seismic load, wind loads and external pressure in its design formula [8]. Therefore, what is needed is design by analysis which requires creativity and action of the designer.

There has been reoccurring cases of gas plants, cylinder explosions across Nigeria, particularly in the LPG domain either during transit, storage or during domestic use [9]. Therefore, there is need to give careful attention to LPG pressure tanks in line with design.

The finite element method is a useful numerical method utilized in solving many engineering problems. Finite element works by breaking down or discretizing a real object/system into a smaller number of finite, well defined sub-structures (element) which can be represented by simple equations [10]. Each of these elements has nodal points, subjected to finite degrees of freedom. The mathematical model developed is formed by assembling all individual elements. The behavior of each element is then used to analyze the performance of the whole system. In applying FEM to any engineering problem, one needs to understand the following: the physical behavior of the system (strength, heat transfer etc.), the performance (safety, weakness), the accuracy of the FEM in comparison to the analytical method [11]. ANSYS is finite element software which allows for visualization of the effect of loads and other boundary conditions on the model been analyzed for easy understanding which does not involve

Writing or interpretation of codes. The results of the analysis can easily be visualized and utilized by local designers/engineers who are not experts in finite element analysis. An ANSYS result, when validated is in harmony with order finite element computational platforms [12, 13].

2. Methodology

ANSYS workbench version 14 finite element computational platform was used in this work.

2.1 Assumptions

- Plain strain condition
- The material selected is homogeneous and isotropic.
- Uniform internal pressure.

The work involved two stages

a. validation of the computational platform to be used
b. Use of 3D finite element model to perform Von-mises stress analysis and displacement in liquefied petroleum gas pressure tanThe work of Oluwole and Emagbetere (2013) was used as bases for validation since similar finite element software (Matlab) was used.

2.2 Finite Element Modeling

Finite element analysis was utilized in this research. The theory of plate elasticity and plate bending was used. When the thickness is small in comparison with other dimensions, the vessels is referred to as membranes and the associated stresses resulting from the contained pressure are called membrane stresses. These membrane stresses are average tension or compression stresses. They are assumed to be uniform across the vessel wall and act tangentially to its surface. The membrane or wall is assumed to offer no resistance to bending. When the wall offers resistance to bending, bending stresses occur in addition to membrane stresses [4].

Membrane element.

$$P = \frac{F}{A} \tag{1}$$

therefore, $F = PA$

where, P is the pressure acting on the inner wall, A is the area, F is the traction force acting on the plate surface.

In order to develop the stiffness matrix and calculate displacements in x and y direction, theory of Elasticity is used [14, 15]. Equilibrium equation in terms of stress is given as;

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + f_x = 0 \tag{2}$$

$$\frac{\partial \tau xy}{\partial x} + \frac{\partial \sigma_y}{\partial y} + f_y = 0 \tag{3}$$

where f_x and f_y are body forces σ_x and σ_y are stress components. The constitutive equation (relating stress to strain) is given as

$$\{\sigma\} = [D]\{\epsilon\} \quad (4)$$

where $\{\sigma\} = \{\sigma_x\ \sigma_y\ \tau_{xy}\}^T$ denotes the stress and $\{\epsilon\} = \{\epsilon_x\ \epsilon_y\ \gamma_{xy}\}^T$ is the strain

If equation (2) and (3) is multiplied with weight function, we have

$$\int_\Omega \begin{Bmatrix} \omega_1\left(\frac{\partial\sigma_x}{\partial x} + \frac{\partial\tau_{xy}}{\partial y}\right) \\ \omega_2\left(\frac{\partial\tau_{xy}}{\partial x} + \frac{\partial\sigma_y}{\partial y}\right) \end{Bmatrix} d\Omega + \int_\Omega \begin{Bmatrix} \omega_1 f_x \\ \omega_2 f_y \end{Bmatrix} d\Omega - \int_{\Gamma e} \begin{Bmatrix} \omega_1 \Phi_x \\ \omega_2 \Phi_y \end{Bmatrix} d\Gamma = 0 \quad (5)$$

The term in the second integral is the body force which is assumed to be zero. While the term in third integral is the traction force which in this case is the force F due to the applied pressure, therefore,

$$F = PA = \int_{\Gamma e} \begin{Bmatrix} \omega_1 \Phi_x \\ \omega_2 \Phi_y \end{Bmatrix} d\Gamma \quad (6)$$

Simplifying equation (5) yields

$$\int_\Omega \begin{bmatrix} \frac{\partial\omega_1}{\partial x} & 0 & \frac{\partial\omega_1}{\partial y} \\ 0 & \frac{\partial\omega_2}{\partial y} & \frac{\partial\omega_2}{\partial x} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} d\Omega = PA \quad (7)$$

Combining equation (4) into (7) gives,

$$\int_\Omega \begin{bmatrix} \frac{\partial\omega_1}{\partial x} & 0 & \frac{\partial\omega_1}{\partial y} \\ 0 & \frac{\partial\omega_2}{\partial y} & \frac{\partial\omega_2}{\partial x} \end{bmatrix} [D] \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \tau_{xy} \end{Bmatrix} d\Omega = PA \quad (8)$$

on further simplification the stiffness matrix is given as;

$$[K^e] = \int_{\Omega e} [B]^T [D][B] d\Omega = [B]^T [D][B] A \quad (9)$$

where [Ke] = [Km] is the element membrane stiffness matrix, [D] is the elasticity matrix and [B] is the strain matrix.

Bending element. For the bending element, we use a three noded plate bending element. Theory of classical plate bending is used [14,16].The displacement function w is assumed to be;

$$\omega(x,y) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 x^2 + \alpha_5 xy + \alpha_6 y^2 + \alpha_7 x^3 + \alpha_8(x^2y + xy^2) + \alpha_9 y^3 = [X]\{\alpha\} \quad (10)$$

where

$$[X] = [1\ x\ y\ x^2\ xy\ y^2\ x^3\ (x^2y + xy^2)\ y^3] \quad (11)$$

$$\{\alpha\} = \{\alpha_1\ \alpha_2\ \alpha_3\ \alpha_4\ \alpha_5\ \alpha_6\ \alpha_7\ \alpha_8\ \alpha_9\}^T \quad (12)$$

Differentiating [X] with respect to x and y gives a 9 x 9 matrix for the three nodes.

Further differentiation per node yields

$$[L] = \begin{bmatrix} 0 & 0 & 0 & 2 & 0 & 0 & 6x & 2y & 0 \\ 0 & 0 & 0 & 0 & 0 & 2 & 0 & 2x & 6y \\ 0 & 0 & 0 & 0 & 2 & 0 & 0 & 4(x+y) & 0 \end{bmatrix} \quad (13)$$

The bending element stiffness matrix $[K_b]$ is given as;

$$[K_b] = [\ddot{X}]^{-T} \int_{\Omega e} [L]^T [D][L]\, d\Omega [\ddot{X}]^{-1} \quad (14)$$

Total element stiffness matrix**.** In order to get the total element (system) stiffness matrix [K], we combine stiffness matrix of the membrane element $[K_m]$ and bending element [Kb] ;

$$[K] = [K_m] + [K_b] \quad (15)$$

The combination takes the following form

$$\begin{bmatrix} K_m & 0 \\ 0 & K_b \end{bmatrix}$$

The finite element equation is expressed as

$$[K]\{U\} = \{F\} \quad (16)$$

where {F} is the applied force, {U} is the displacement.

2.3 Von-Mises Stress

For the Von-mises stress to be calculated analytical, there are three principal stresses which are;

σ1 = Principal stress = Longitudinal (axial)stress

σ2 = Principal stress = Circumferential (hoop) stress

σ3 = Radial stress = 0. No stress in z-direction that will lead to displacement or elongation.

Von-mises stress

$$= \sqrt{\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2} \quad (17)$$

$$\sigma 2 = \frac{Pr}{2t} \quad (18)$$

$$\sigma 1 = \frac{Pr}{t} \quad (19)$$

p = internal pressure
r = radius of cylinder
t = plate thickness

2.4 Factor of Safety (FOS)

The material already has a factor of safety of 3.5, therefore, for each simulation carried out per tank plate thickness, the factor of safety is calculated to determined safety of the vessel at that operating pressure. For this research work, the factor of safety is calculated as follows:

$$FOS = \frac{\text{Material minimum tensile strength}}{\text{Material Allowable stress}} \quad (20)$$

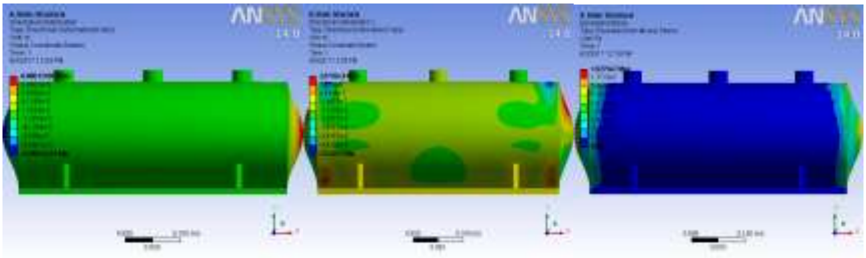
Material Allowable stress = Finite element Analysis Von-Mises Stress (equivalent stress developed during simulation with ANSYS static structural)

3. Validation of the Finite Element Computational Platform

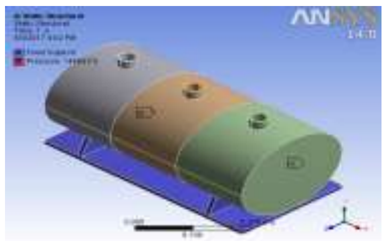
Finite element analysis of displacement and Von-mises stress in pressure vessel has already been done with a case study in petroleum road tankers. The tank content is diesel (AGO), with a loading pressure of 14480 N/m2 The analysis was done using Matlab programming. This work did not consider the effect of increasing pressure at elevated temperature on the tank plate thickness. Also the contour plotting are line plots and requires interpretation of written codes to visualize the effects of loads and other boundary conditions. To validate this work, ANSYS static structural was used with the same material properties and simulation parameters as used in Matlab.

3.1 Parameters Used for Validation

Length of tanker = 485 cm
Vertical axis of tanker = 180 cm
Horizontal axis of tanker = 244 cm
Thickness of tanker = 0.2 cm
Poison ratio = 0.3
Loading pressure = 14480 N/m2
Material of construction = A516M Grade 70
Specified minimum yield stress = 25 × 107 N/m2
Maximum allowable stress = 13.8 × 107 N/m2
Elastic modulus = 200 × 109 N/m2



(a) (b) (c)



(d)

Fig. 1 ANSYS Static Structural Validation for Diesel Tanker (a) Displacement in x-axis (b) Displacement in y-axis (c) Von-Mises Stress. (d) Tank model before simulation

Figure 1 above shows that the Von-Mises stress is tensile in nature, causing the elliptical section of the tank to bulge out. Areas in the contour plotting shown in red are areas where the Von-mises stress is mostly felt, hence these areas will experience more displacements. The result in comparism with Matlab is shown in the table below.

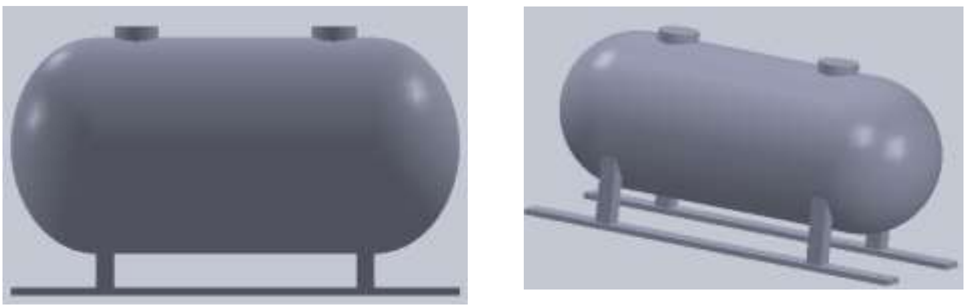
Table 1 Camparism of Matlab generated result with ANSYS Static Structural for validation of a diesel tanker.

FEA Application	Displacement in x-axis (m)	Displacement in y-axis (m)	FEAVon-Mises Stress N/m^2	ASME Von-Mises Stress N/m^2
Matlab Program	5.2201×10^{-9}	1.4789×10^{-7}	5.4318×10^{6}	7.6494×10^{6}
ANSYS Static Structural	9.6507×10^{-5}	2.0716×10^{-6}	6.425×10^{6}	7.6494×10^{6}

As shown in the table above, the result of the Matlab program is much identical to that of ANSYS Static Structural. In fact, the FEA Von-Mises Stress of ANSYS Static Structural is in close range with the ASME Von-Mises stress that is the analytical Von-mises. Having validated the result, the research work proceeded with the application of ANSYS Static Structural for the finite element analysis of liquefied petroleum gas pressure tank model.

3.2 Development of the LPG Cylindrical Pressure Tank Model for Simulation

In order to reduce computational complexities, the LPG tank model was made simple. The cylindrical pressure tank model (Fig. 2) was developed into different thicknesses: 2mm, 5mm, 10mm, 20mm and 30mm using Solidworks. Each of this model was imported into ANSYS static structural analysis system independently and the simulation was carried out in this sequence; Analysis system (static structural), Engineering Data, Geometry, Model, Setup and Solution**.**



(a) (b)

Fig. 2 Views of the Model of the LPG Pressure Tank

3.3 Statics Analysis for the LPG Tank

This involves application of finite element analysis include meshing, boundary conditions and the material properties specification etc.

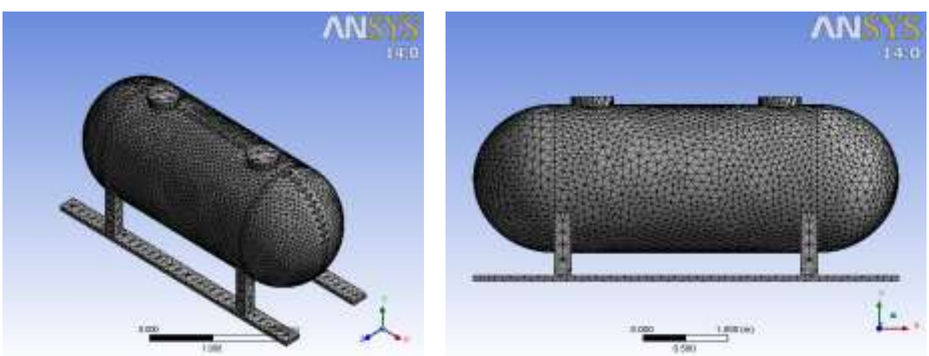


Fig. 3 meshing at (a) 2mm plate thickness and (b) 10mm plate thickness

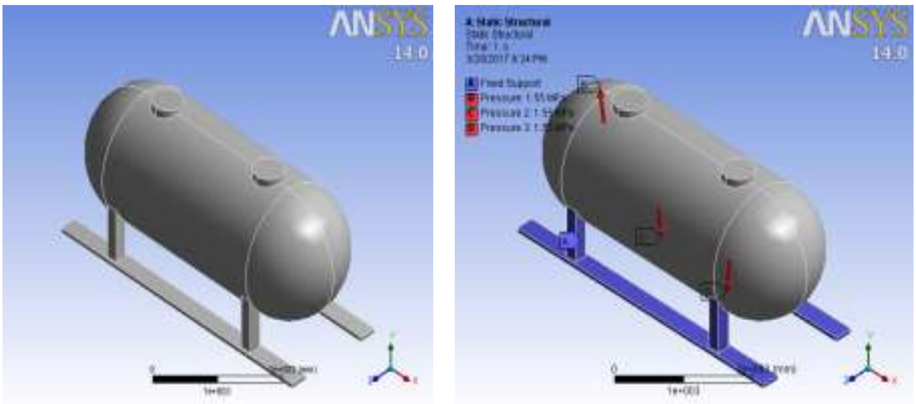
Meshing: Meshing is critical to any modeling and simulation work. For the LPG tank, the mesh size chosen was fine mesh and the smoothing was medium. This was done to influence the accuracy and the computing speed. As plate thickness increases, number of nodes and elements increases. Figure 3 is a view of the different kinds of mesh utilized in this work.

Boundary condition: In this part of the simulation

(a)

(b)

, the boundary conditions are specified. The internal pressure applied are 0.5MPa, 0.91MPa and 1.55MPa each at different plate thickness and ambient temperature: 20^0C, 40^0C and 60^0C respectively. The base of the vessel is fixed to a support (dirichlet boundary condition).There are two in-plane displacement u and v in x and y directions and one deflection w in z-direction.



(a) (b)

Fig. 4 (a) Tank model imported to ANSYS Static Structural (b) Application of boundary condition

3.4 Tank Parameters for Analysis

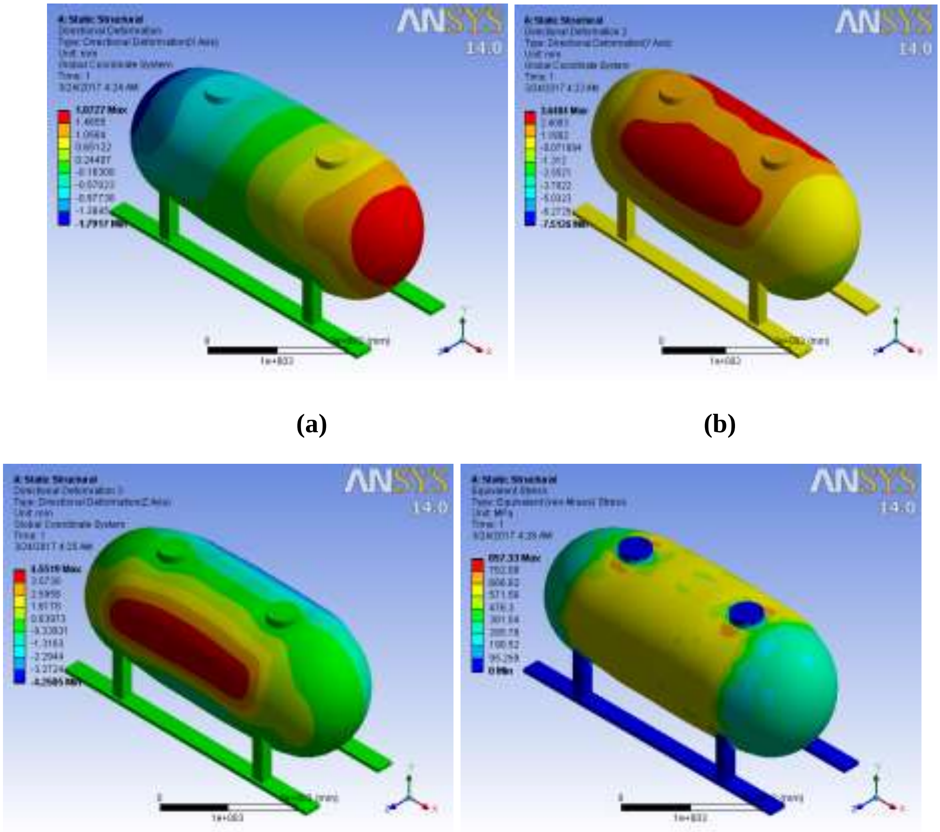
Length of tank = 495cm
Internal diameter = 190cm
Diameter of head = 95cm
Plate thickness = 2mm, 5mm, 10mm, 20mm and 30mm. (These range of thickness are in line with ASME SECTION VIII DIVISION 1 PART ULT).
Tank material: ASTM A516 Grade 70
Material allowable stress = 138MN/m^2
Material minimum yield stress = 260MN/m^2
Material minimum tensile strength= 485MPa
Modulus of elasticity = 200GN/m^2
Material factor of safety = 3.5

4. Simulation of the Liquefied Petroleum Gas Pressure Tank

The simulation was carried out in stages as highlighted below:

4.1 Simulation at 60^0C, 1.55MPa (Case 1)

The tank parameters for analysis are as stated above. Each cylindrical LPG pressure tank model of thickness: 2mm, 5mm, 10mm and 30mm was subjected to same internal pressure and temperature.
LPG Temperature = 60^0C
Internal pressure = 1.55MPa



(a) **(b)**

(c) **(d)**

Fig. 5 Application of 1.55MPa at 60^0C to 2mm tank model thickness (a) displacement in x-axis (b) displacement in y-axis (c) deflection (d) Von-Mises stress

4.2 Results and Discussion as Presented in Case 1

Results. Figure 5 (a) and (b) shows the displacement in x and y direction. The contour plotting in red are areas where the displacement is more pronounced. This is similar to the deflection as shown in Fig. 5 (c). ANSYS Von-mises stress causes the head of the tank to enlarge/bulge out and the deformation of the cylindrical section as shown in Figure. 5 (d). Table 2 shows the displacement and deflection at different plate thicknesses. As plate thickness increases, displacement in x and y direction and deflection in z decreases. This is pictorially illustrated in Figure. 6, Figure 7 and Figure. 8. Also, the Von-mises stress converges to zero as the plate thickness increases as seen in Figure. 9. Table 3 shows the variation in Factor of safety, at different ASME and FEA stresses and plate thicknesses.

Table 2. Displacements and deflection at different plate thickness for cylindrical LPG pressure tank at 1.55MPa, 60^0C

PLATE THICKNESS (mm)	DISPLACEMENT IN X-AXIS (mm)	DISPLACEMENT IN Y-AXIS (mm)	DEFLECTION (Z-AXIS) (mm)
2	1.87270	3.64840	4.55190
5	0.79979	1.27340	2.36770
10	0.39974	0.61834	1.12540
20	0.19751	0.34636	0.49299
30	0.12950	0.24184	0.28865

Table 3 ASME stress, FEA stress and Factor of Safety at different plate thickness for LPG at 1.55MPa, 60^0C

PLATE THICKNESS (mm)	FEA Von-Mises/Stress developed (MPa)	ASME Von-Mises stress (MPa)	(FEA) Factor of Safety
2	857.33	637.61	0.57
5	401.02	254.00	1.21
10	208.71	127.00	2.32
20	102.91	63.77	4.71
30	64.34	42.50	7.54

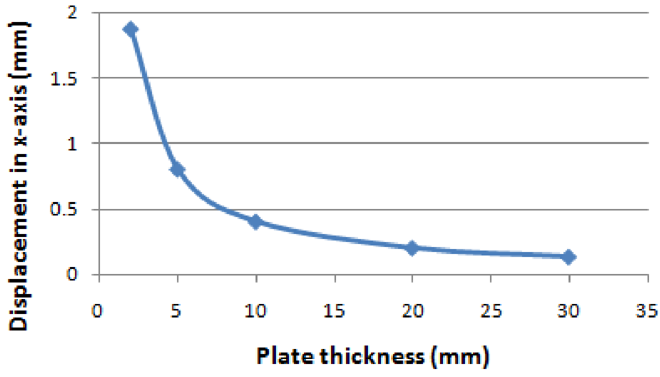


Fig. 6 In plane displacement in x-axis versus thickness at 1.55MPa, 60^0C

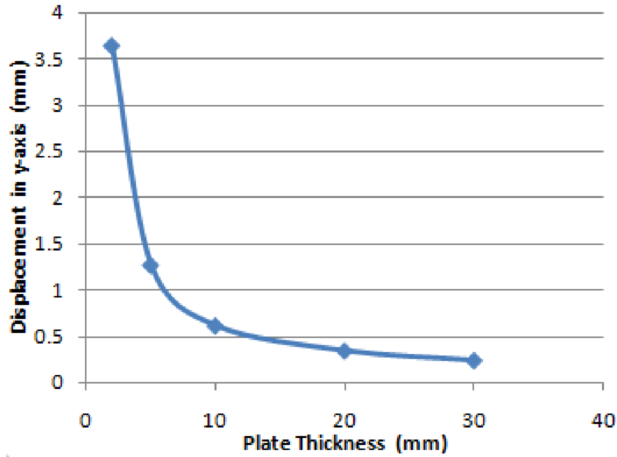


Fig. 7 In plane displacement in y-axis versus thickness at 1.55MPa, 60^0C

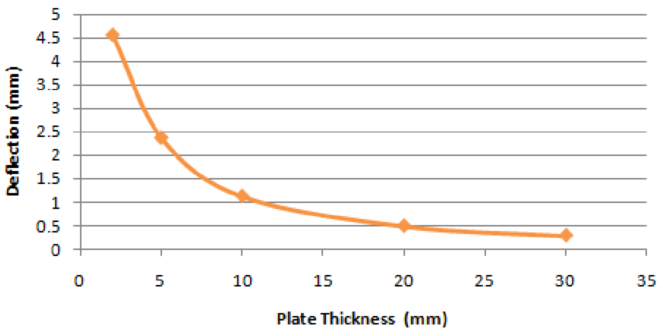


Fig. 8 Deflection versus thickness at 1.55MPa, 60^0C

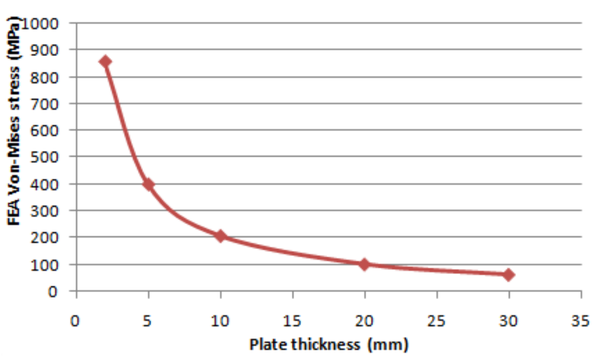


Fig. 9 FEA Von-Mises Stress (stress developed)Versus thickness at 1.55MPa, 60^0C

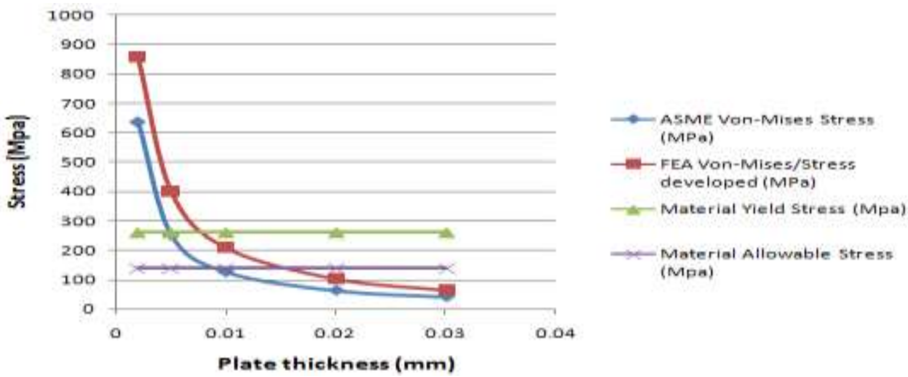


Fig. 10 Comparing FEA Von-Mises (stress developed) at 1.55MPa, with ASME Von-Mises stress, Material yield stress and Allowable stress.

Discussion of Results as Presented in Case 1. Increasing the LPG pressure tank plate thickness decreases the displacement, deflection and Von-mises stress as presented in Fig. 6, 7, 8 and 9. The plate material (ASTM A516 Grade 70) of the LPG tank already have a factor of safety of 3.5. For the range of thicknesses considered as shown in Table 3, 10mm thickness and below will cause catastrophic failure if the LPG pressure tank is to operate at 1.55MPa and 60^{0}C since there factor of safety is less than 3.5 (material's factor of safety). At 20mm thickness and above, the tank material will not yield (failure will not occur) since this range of thickness offers factor of safety greater than 3.5. Considering Fig.10, the graph of material allowable stress intersects the graph of FEA Von- mises stress (stress developed) at about 15mm. Therefore, 15mm could be taking as the minimum plate thickness for LPG pressure tank operating at 1.55MPa and 60^{0}C. Since the vessel material is isotropic in nature, increasing plate thickness will keep the hoop stress/circumferential stress below the material yield stress, therefore, it will be twice as strong in the axial direction. The major disadvantage is the increase in weight of the vessel.

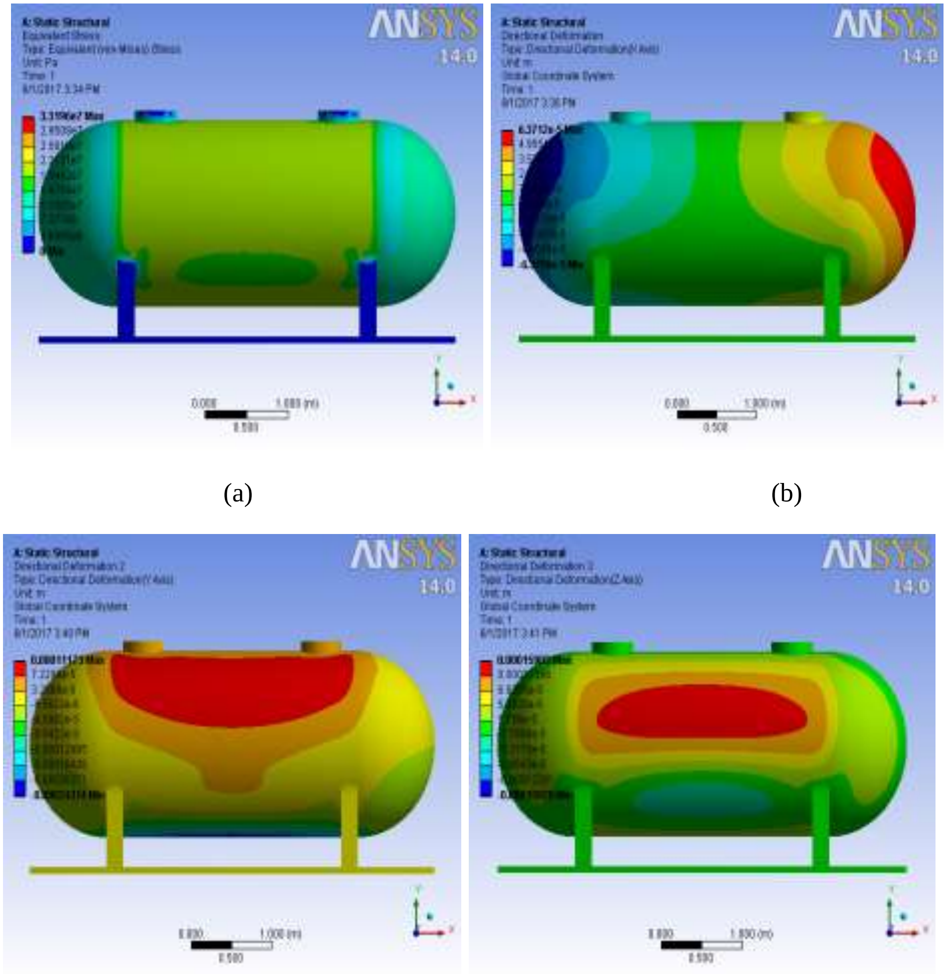
4.3 Simulation at 20^{0}C, 0.5MPa (Case 2)

For cylindrical LPG pressure tank model of thickness: 2mm, 5mm, 10mm and 30mm each subjected to same internal pressure and temperature

LPG Temperature = 20^{0}C

Internal pressure = 0.5MPa

The tank parameters for analysis are the same as in case 1and 2 except the temperature and LPG pressure.



(a) (b)

(c) (d)

Fig. 11 Application of 0.5MPa at 20^{0}C , 10mm thickness (a) Von-Mises stress (b) displacement in x-axis (c)displacement in y-axis (d) deflection (z-axis).

4.4 Results and Discussion as Presented in Case 2

Results. Figure 11 shows the ANSYS static structural contour plots of the LPG pressure tank at 0.5MPa and 20^{0}C. Figure 11 (b) and (c) shows the displacement in x and y direction while (d) shows the deflection in z direction. Plots in red are area where the biaxial state stress is mostly felt. These results are presented in tabular form as shown in Table 6. Graphical presentation of these results is similar to Fig. 6,7 and 8. Table7 follow the same trend as presented in Table 3 of case 1.

Table 6 Displacements and deflections at different plate thickness for cylindrical LPG pressure tank at 0.5MPa, 20^{0}C

PLATE THICKNESS (mm)	DISPLACEMENT IN X-AXIS (mm)	DISPLACEMENT IN Y-AXIS (mm)	DEFLECTION (Z-AXIS) (mm)
2	0.604060	1.176200	1.467600
5	0.258000	0.410780	0.763770
10	0.063712	0.111730	0.159030
30	0.041815	0.077964	0.092854

Table:7 ASME Stress, FEA stress and Factor of Safety at different plate thickness for LPG at 0.5MPa, 20^{0}C

PLATE THICKNESS (mm)	FEA Von-Mises/Stress developed (MPa)	ASME Von-Mises stress (MPa)	(FEA) Factor of Safety
2	276.61	205.68	1.75
5	129.36	82.28	3.75
10	33.196	41.14	14.61
30	20.722	13.71	23.41

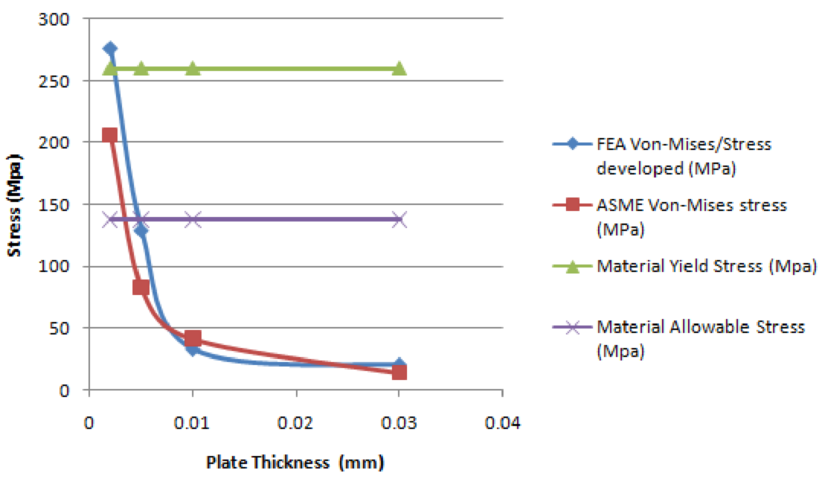


Fig. 12 Comparing FEA Von-Mises (stress developed) at 0.5MPa with ASME Von-Mises stress, Material yield stress and Allowable stress.

Discussion of results as presented in simulation case 2. Displacement, deflection, Von-mises stress and the factor of safety follow the same trend as simulation case 1. The FEA Von-mises stress that is the stress developed shows some correlation with the ASME Von-mises stress. In Table 6, the deflections are more than displacement values since the hoop stresses often results to bending of the vessel plate material. Considering Table 7, at 5mm thickness, the finite element factor of safety (3.75) is greater than the material's factor of safety (3.5). Also, this is illustrated graphically in Figure 12 in which the graph of material allowable stress intersets the graph of FEA Von-mises stress at 5mm. Therefore, it can said that at LPG pressure of 0.5MPa and ambient temperature of 20^0C, the minimum plate thickness recommended is 5mm.

4.5 Simulation at 40^0C, 0.91MPa (Case 3)

The same range of thickness was maintained (2mm, 5mm, 10mm, 20mm and 30mm), tank material properties remains the same but operating temperature and pressure was changed.

LPG Temperature = 40^0C
Internal pressure = 0.91MPa

Table 4 Displacements and deflection at different plate thickness for cylindrical LPG pressure tank at 0.91MPa, 40^0C

PLATE THICKNESS (mm)	DISPLACEMENT IN X-AXIS (mm)	DISPLACEMENT IN Y-AXIS (mm)	DEFLECTION (Z-AXIS) (mm)
2	1.099400	2.14060	2.67100
5	0.470400	0.74904	1.39510
10	0.234690	0.36303	0.66072
20	0.115960	0.20335	0.28944
30	0.076102	0.14189	0.16899

Table 5 ASME stress, FEA stress and Factor of Safety at different plate thickness for LPG at 0.91MPa, 40^0C

PLATE Thickness (mm)	FEA Von-Mises/Stress developed (MPa)	ASME Von-Mises stress (MPa)	(FEA) Factor of Safety
2	503.430	374.34	0.96
5	235.480	149.74	2.06
10	122.530	74.87	3.96
20	60.417	37.44	8.03
30	37.714	24.96	12.86

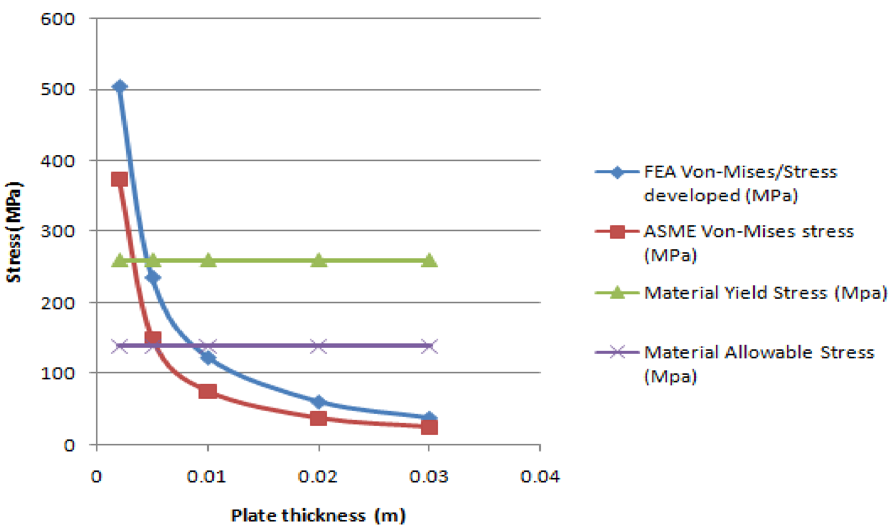


Fig. 13 Comparing FEA Von-Mises (stress developed) at 0.91MPa with ASME Von-Mises stress, Material yield stress and Allowable stress.

4.6 Results and Discussion as Presented in Case 3

Table 4 and 5 follows the trend of case 1 and 2. Figure 13 shows the non linear relationship between stress and plate thickness. It also shows the convergence of finite element Von-mises stress(stress developed) and ASME Von-mises stress. The inverse relationship between thickness and stress is due to the disparity between circumferential stress (hoop stress) and plate thickness. Since the tank material is usually welded, therefore the welded area experience HAZ (heat-affected-zone). As pressure increases, hoop stress builds up in the heat-affected-zone, leading to crack initiation, propagation and material failure. This will occur once the stress developed is above the material allowable stress. For the range of thickness considered, 10mm thickness is taking as the minimum plate thickness at 0.91MPa since it offers factor of safety greater than the material's factor of safety. In Figure 11, the graph of material allowable stress intersects the graph of FEA Von-Mises stress at 10mm thickness showing that failure will not occur at this thickness and above it.

5. Conclusion

The Von-Mises stress and displacement in the Liquefied petroleum gas (LPG) pressure tank under different pressure distribution and ambient condition has been obtained using the finite element method. As temperature increases, LPG pressure increases, hence, there is need to design the pressure tank in such a way that the thickness will accommodate the rise in pressure. This will yield better results and reduce the risk of an explosion. For the different pressure range considered: 0.5MPa, 0.91MPa and 1.55MPa, the Von-Mises stress decreases with increasing plate thickness. A minimum plate thickness was deduced for each pressure range: 5mm thickness for 0.5MPa, 10mm thickness for 0.91MPa and 15mm thickness for 1.55MPa. At this minimum plate thickness, the Von-Mises stresses were found to be lower than the tank material allowable stress ($138MN/m^2$). The finite element Von-mises stress developed during simulation were in the same range with the ASME Von-mises. The range of thickness and stress are in compliance with ASME section VIII division 1 part ULT. The vessel material ASTM A516 Grade 70 already has a factor of safety of 3.5; therefore, design consideration should include material's yield and allowable stress and factor of safety greater than 3.5. For this research work, there are different possible scenarios. Once the boundary condition changes, the result will change, therefore, each should be treated as a case study. The effect of weldment along the seams of the vessel was not carried out in this work.

References

[1] Foraminifera Market Research. Liquefied Petroleum Gas Bulk Storage and Marketing in Nigeria; How Viable? Retrieved February 10th 2017 from

http://www.foramfera.com/liquefied- petroleum-gas-bulk-storage-and-marketing-in-nigeria-how-viable/. (2017)

[2] Bruce G. The Smell of Danger. Chemmatters. Journal of American Chemical Society.Retrieved February 2nd 2017 from http://brucegoldfarb.com/clips/GoldfarbPropane.pdf. (1988).

[3] Abdul-Kadir K. A. Domestic LPG Market Growth-Infrastructural Challenges and Opportunities. Retrieved 02/02/17 from http://nigerialpgas.com/downloads/Domestic_LPG_Market_Growth_Infrastructural_Challenges_and_Opportunities.pdf. (2016).

[4] Dennis R.M. Pressure Vessel Design Manual: Illustrated Procedures for Solving Major Pressure Vessel Design Problems, New York: Gulf Professional Publishing. pp. 10-109. (2004).

[5] Dražan K., Ivan S., Antun S., Željko I. and Darko D. Stress Analyses of Cylindrical Vessel with Changeable Head Geometry. Scientific Bulletin, Series C, Volume XXIII, Fascicle: Mechanics, Tribology, Machine Manufacturing Technology. ISSN 1224-3264. (2009).

[6] Adeyefa O. and Oluwole. O. Finite Element Modeling of Variable Membrane Thickness for FieldnFabricated Spherical (LNG) Pressure Vessels. http://www.scirp.org/journal/eng. (2013).

[7] ASME. ASME Boiler and Pressure Vessel Codes. The American Society of Mechanical Engineers, New york. Library of Congress Catalog Card Number 56-3934. (2004).

[8] Oluwole O and Emagbetere E. Finite Element Analysis of In-plane Displacements and Von-Mises Stresses in Ellipsoidal and Circular Cylindrical Petroleum Tankers. (http://www.scirp.org/journal/eng). (2013).

[9] Awoyinfa S. Two killed, seven injured in Ogun gas plant explosion. Punch Newspapaer Nigeria. punchng.com/two-killed-seven-injured-ogun-gas-plant-explosion/. (2017).

[10] Richard G.B. Advance Strength and Applied Stress Analysis. Second edition. McGraw hill publishing companies Inc. (1999).

[11] Adeyefa O.A. Finite Element Analysis of Double-Jacked Field-Fabricated Spherical Liquefied Natural Gas (LNG) Pressure Vessels. Ph.D. Thesis, University of Ibadan (2015).

[12] Jorge R.M., Helder S.W. and Carlos A.C. Stress Analysis on Vessel/Nozzle Intersections With/Without Pad Reinforcement for Cylindrical Pressure Vessels. Proc. of 19th Int Congress of Mech. Engineering, Brasília, DF. (2007).

[13] Morrish K. and Shankar K.M. Comparative Stress Analysis of Elliptical and Cylindrical Pressure Vessels With and

Without Autofrettage Consideration Using Finite Element Method. Int. J. Adv. Engg. Res. Studies/IV/II/, pp. 189-195 (2015).

[14] Young W.K. and Hyochoong B. Finite Element Method Using Mathlab, New York: CRC Press, pp. 307- 373. (1997).

[15] Timoshenko T. and Goodier J. Theory of Elasticity, New York: McGraw Hill Books Company Inc., pp. 255-257.(1951)

[16] Zienkiewicz O.C. and Taylor R.L. The Finite Element Method, Oxford: Butterworth-Heinemann, vol. 2, pp. 111-210. (2000).