NONLINEAR FE ANALYSIS OF SITU QUARTER DIESEL TANK UNDER PRESSURIZED LOADING CONDITION

[1]Syed Faisal Hussaini,[2]Ambadas,[3]Babu Reddy
[1]M.Tech Student,[2& 3]Assistant Professor
[1, 2 & 3]Department of Machine Design, VTU Center for PG Studies,Kalaburagi, India

Abstract—The main objective of this paper is to design a quarter diesel tank and carry out nonlinear FE Analysis under Pressurized loading condition. The diesel tank designed mainly for placing underground covered with the soil levels. The purpose for designing the tank is due to some cases of fire burst explosions, collisions, leakage etc, so tank safety is the major concern. In this analysis our aim is to predict the maximum stresses as well as deformations under the nonlinear characteristics of the components. On the basis of the detailed FEA using Ansys software, prediction of performance of the entire diesel tank is preferred. Boundary conditions considered are as per the actual operational environment.

IndexTerms—Nonlinear Analysis, Pressurized loading, Stresses and Deformations, FEA, ASME Standards etc.

I. INTRODUCTION

Underground tank as a component of ecological building offices and principally utilized for fuel and sewage treatment plants and other mechanical uses. They are used for various purposes and are important component for holding the liquid. The tanks are made of various materials. They are either made of strengthened cement as rectangular or round arrangements of soils or by using various materials available on earth using different combinations. Even now the tanks are made of plastics. So with the underground tanks the major concern is bursting or explosion due to storage of liquids of flammable nature.

Storage tanks in refineries and chemical plants contain large volumes of flammable and hazardous chemicals. A small accident may lead to huge property loss and production interruption as well. A decent comprehension of the seismic conduct of these structures is vital keeping in mind the situation that can happen due to this. In this paper underground tank placed under the soil is considered for valuation. The Finite Element Method (FEM) is chosen as the examination strategy for the underground tank. In this research a standard underground tank as per ASME standards is to be analyzed under pressurized loading conditions. For our case nonlinear analysis of the situ tank is to be done, non linearity is by geometry of the tank. The analysis is carried out on FEA software package ANSYS and meshing of tank carried out in Hypermesh. In this analysis our aim is to predict the maximum stresses as well as deformations under the nonlinear characteristics of the components.

In this work a 3D CAD of situ quarter tank is considered to analyze its capacity under pressurized loading conditions. This is a special purpose tank of high strength and durability, so this tank will be placed under different soil levels. The load applied by the soil on the tank is the factor used to determine the tank performance. The load applied is by considering the worst conditions.

II. LITERATURE REVIEW

A CAD of actual size, considering half-symmetry, underground storage tank model is investigated inside two FEA packages, in particular ANSYS and ABAQUS.[1] Underground water tank with various safe bearing estimations of soil is utilized to examine the common conduct caused by seismic pressure on the soil.[2] Consider the impact of different parts of seismic tremor on sloshing reaction of storage tanks. They considered unidirectional examination of fluid in cylindrical tanks.[3] The optimum seismic design was achieved satisfying the stability of these tanks under extreme seismic design loads according to the Eurocode or the Greek seismic regulation provisions. This method gives the most economical dimensions for the tank and its foundation, for a predefined, design liquid volume.[4] the accidents related to storage tanks due to various reasons were given. Maximum cases were from oil and gas refineries where the storage tanks are placed underground, fire and explosion during the maintenance works, burst due to internal pressure. [5]

III. OBJECTIVE AND METHODOLOGY

3.1 Objectives Analyze the CAD model of Quarter tank, Tank is underground storage type placed under different soil layers, CAD model is prepared in CATIA software, Meshing is done in HyperMesh, Nonlinear FE analysis is carried out in ANSYS to determinedeformations in x, y and z directions, Maximum Principal Stress, von Mises Stress and Strain.

3.2 Methodology The CAD model of the tank is to be designed in CATIA software and save in step file format to exchange the file with other softwares. The CAD file is to be imported in the HyperMesh software. Meshing is done and material properties are assigned in HyperMesh because it generates high quality mesh in very less time even for a complex model. ASME Pressure Vessel Code is referred for the selection of materials for tank components. For the analysis the worst conditions are employed based on available data. The meshed model is then imported in the ANSYS software. ANSYS is used for solving the model by applying the boundary conditions and load type. The results obtained are deformations in x, y and z direction, stresses and strains. After FE analysis the results obtained are compared with the results of experimental data and check the values obtained are within the specified limits and finally conclude with the verdict whether the tank design is safe or not.

IV. QUARTER TANK MODEL, DESIGN CODE, MATERIAL SELECTION

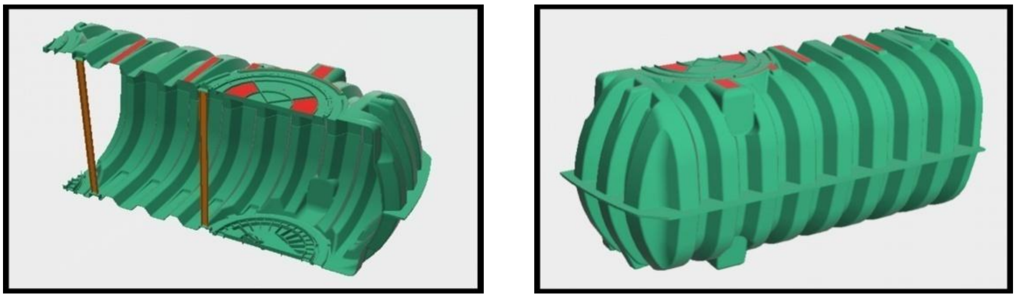


Fig.1 Quarter Tank Model

4.1 Design Code

The following code shall form the basis for design, testing and acceptance of storage tank.

- ASME Boiler and Pressure Vessel Code 2011A Addenda, Section VIII Division 1 and 2: Rules for Construction of Pressure Vessels

4.2 Selection of Materials

All the materials selected are as per the ASME Section VIII Div 1 & 2 Code

- The material for tank is A216 grade WCB SI LF6 which is a Wrought Carbon Steel Casting. It has Young's Modulus of 160 ksi, density of 0.034 lb/in^3 and has a Yield strength of 350 MPa
- The material for pin is MZ 316 GR 51B l which is a Stainless Steel; it has 2 to 3% Molybdenum for corrosion resistance. It has Young's Modulus of about 350 ksi, density of 0.034 lb/in^3 and has a Yield strength of 650 MPa
- The material used for cover is GR 410 low shade WCB which is a martensitic stainless steel; it contains 11.5% Chromium. Its Young's Modulus is 210 ksi, density 0.032 lb/in^3 and Yield strength of 320 MPa
- For external support walls the material is of high strength having ultimate tensile strength of up to 1035 MPa

V. FE ANALYSIS OF QUARTER TANK

The basic idea of FEA is to minimize the infinite solution to finite solution by decreasing the number of nodes and elements. This can also lower the complications of problem solving and get the results in less time. This can happen only when the component is discretized properly according to the shape of the component. Meshing is the main stage in analysis. Selection of type of element isimportant. In our case 3D meshing is done using tetrahedron elements. The tetrahedron element is used for tank part. Meshing was carried out in HyperMesh software.The CAD model is imported in the HyperMesh software. The geometry of the model is carefully examined before starting meshing because if meshing is not proper the results may not be accurate. According to geometry of the model the 3D meshing is done by selecting tetrahedron element solid 92 for the tank part. The meshed part is as shown in the figure 2. After meshing, material properties and boundary conditions are assigned to the tank component.

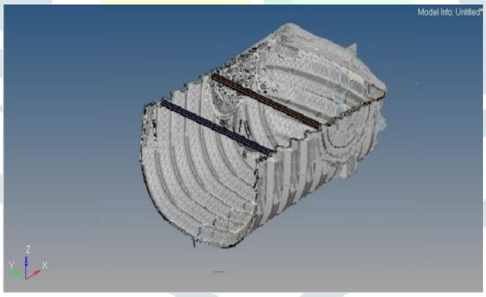


Fig.2 Meshed Model in HyperMesh

Total No of Nodes: 14 03 176 and Total No of Elements: 12 82 345

This meshed model after assigning material properties and boundary conditions is saved for ANSYS software. The model is imported in ANSYS for solving. In ANSYS following procedure is carried out. From the options select Solution and then Post processing.

Solution:

- Select the analysis type
- Define number of load steps and sub steps
- Define the solver
- Define the output options
- Solve

Post processing:

- Read the load step from the result files
- Check the reactions forces at supports
- Plot the displacements, stresses, strain, etc
- Conclude the results

5.1 Nonlinear Analysis of Quarter Tank:

Nonlinear analysis of the tank model is done because of geometrical nonlinearity. Tank model is of irregular shape so linear analysis does not give accurate results and plastic deformation cannot be identified so nonlinear analysis is chosen. The tank model is given boundary conditions and pressure loading is applied as shown in figures below. Bottom face of the model is fixed in x and y direction, red color faces A and B are fixed in z and x direction. The pressure load of 1.6 psi is applied; this load is considered for the worst conditions of tank operation.

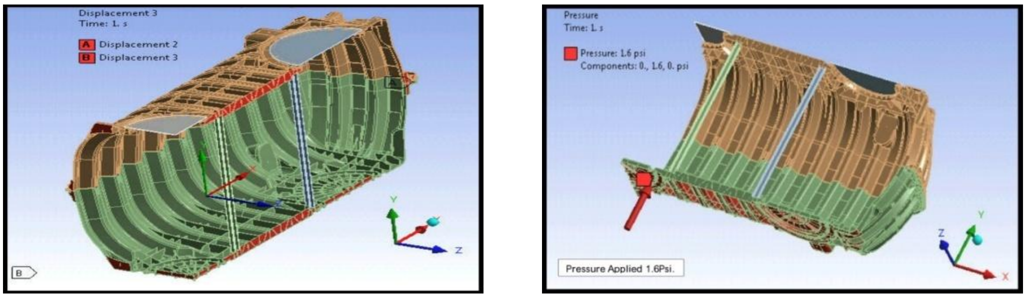


Fig.3 Boundary Condition and Loading

After applying boundary conditions as shown in above left picture, pressure load is applied on the surface of the tank as shown in right picture above. The results are obtained after solving the model. The plots are obtained for deformations in x, y and z directions, vector deformation, maximum principal stress, von Mises stress and strain. The plots are as shown below. The red portions in the plot show the critical part where maximum stress is occurring.

5.2 Plot of Deformations, Stress And Strain

1. Deformation in X direction:

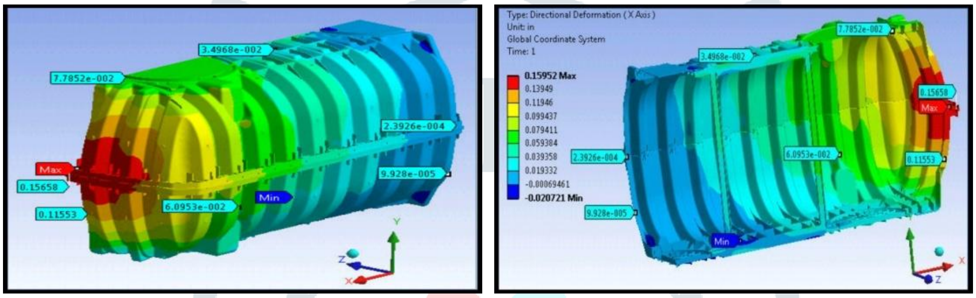


Fig.4 Maximum Deformation is 0.1595 in

2. Deformation in Y direction:

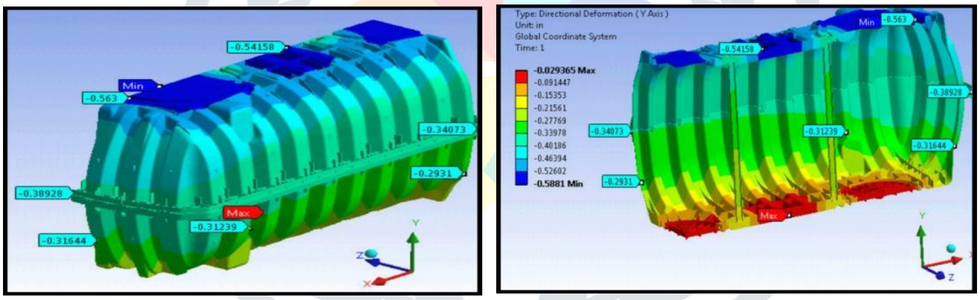


Fig.5 Maximum Deformation is 0.5881 in

3. Deformation in Z direction:

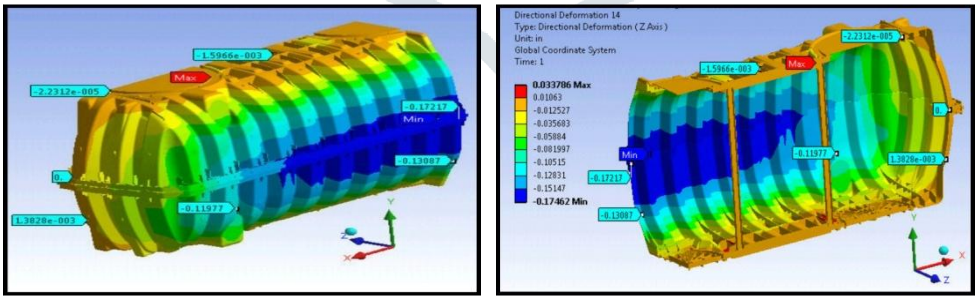


Fig.6 Maximum Deformation is 0.1746 in

4. Vector Deformation:

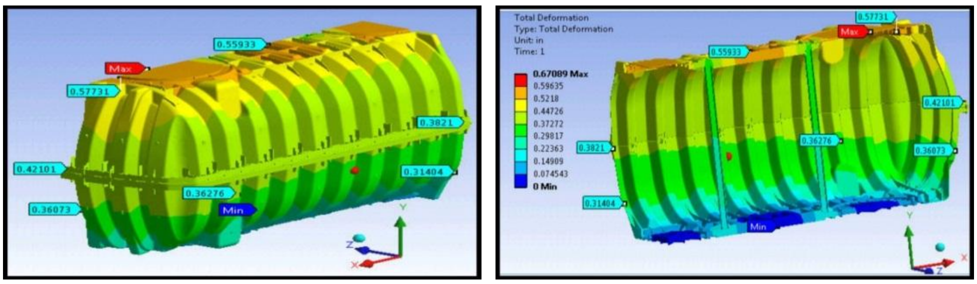


Fig.7 Maximum Deformation is 0.6709 in

The deformations in all three directions are shown in above plots. We can notice that the maximum deformation is in Y direction, as it is the direction where the load is acting.

5. Maximum Principal Stress:

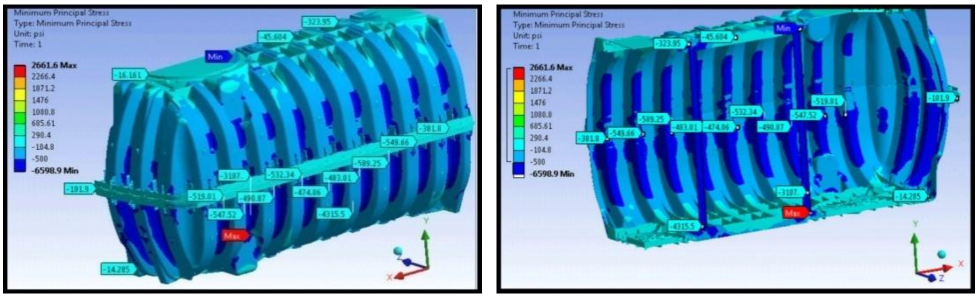


Fig.8 Maximum Stress is 6598.9 psi = 45.49 MPa

6. von Mises Stress:

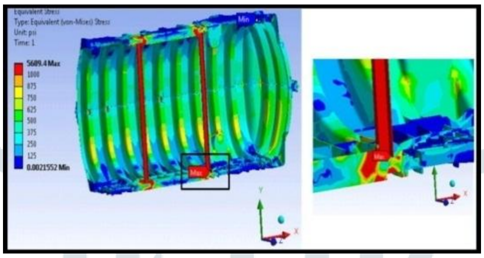


Fig.9 Maximum Stress is 5689.4 psi = 39.22 MPa

7. Maximum Equivalent Strain:

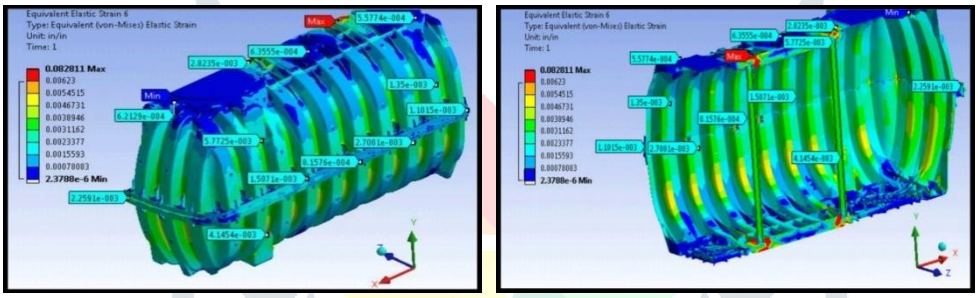


Fig.10 Maximum Strain is 0.0828

5.3 FEA Results

Sl. No	Results	Values
1	Deformation in x direction, δ_x	0.1595 in
2	Deformation in y direction, δ_y	0.5881 in
3	Deformation in z direction, δ_z	0.1746 in
4	Total Deformation, δ_T	0.6708 in
5	Maximum Principal Stress, σ_1	45.49 MPa
6	von Mises Stress, σ_{von}	39.22 MPa
7	Maximum Equivalent Strain, ε	0.0828

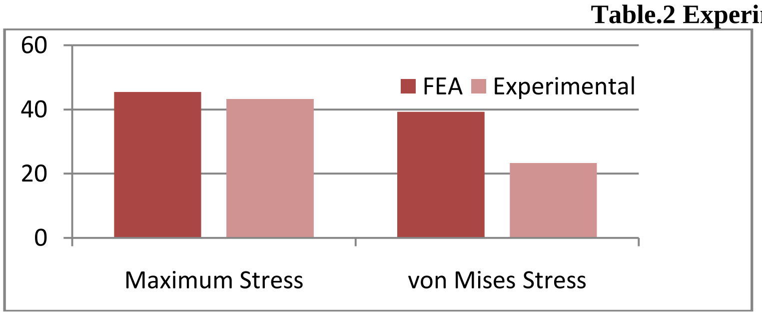
Table.1 FEA Result Tabulation

VI. FEA RESULT CORRELATION WITH EXPERIMENTAL RESULTS

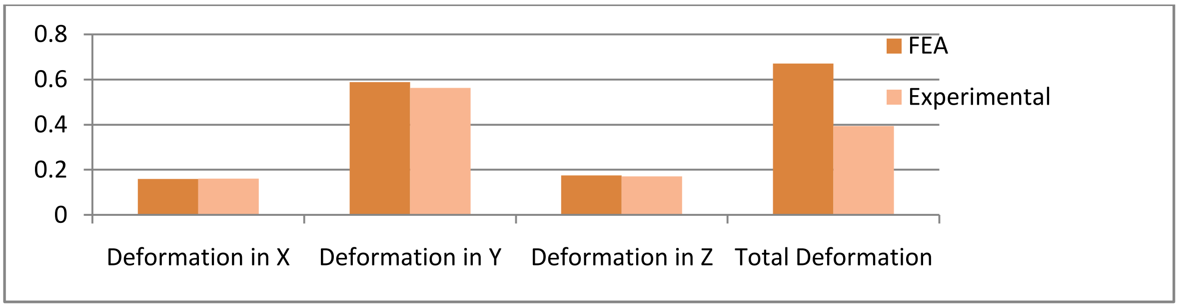
The tank was tested by burying in the ground after complete test setup established as per required operating condition. The pressure was applied on the tank surface and strains were determined in different directions. The test results are as mentioned in Table 2.

Load Case	δ_x	δ_y	δ_z	δ	σ_1	σ_{von}	Comments
Unit	in	in	in	in	MPa	MPa	-
P=1.6 psi	0.1602	0.5632	0.1702	0.3933	43.23	23.34	Values within limit

Table.2 Experimental Results



Graph.1 Stresses Comparison



Graph.2 Deformations Comparison

VII.CONCLUSION

We have successfully designed the tank model considering all the limitations and conditions and as per ASME standards. The maximum stress is acting at the connection of tank and vertical support which can be seen with red color in the plot of von Mises stress. So the critical part of the tank is identified and it is the vertical support. The maximum stress is acting at the connection area. As from the observations, von Mises stress obtained in FEA and experiment is within the limit of yield strength that is 39.22MPa, 23.34 MPa< 350 MPa. The deformations obtained are appropriate as per the selected criterion. The Maximum principal stresses are within the limit of material strength that is 45.49 MPa, 43.23 MPa<350 MPa. So the design is safe.

REFERENCES

[1] Si-Chung Jong, Kim Yeow Tshai, "Finite Element Analysis of Underground Storage Tank Subjected to External Loadings", IJIMM Volume 1, Issue 2, 17th July 2014

[2] Punith C, Usha, Jayalakshmi Raju, M. Adil Dar "Finite Element Analysis of Underground Water Tank with different Safe Bearing Values of Soil", SSRG-IJCE-EFES April 2015

[3] Anumod A. S, Harinarayanan S, S. Usha "Finite Element Analysis of Steel Storage Tank Under Siesmic Load", IJERA ISSN: 2248-9622 Trends and Recent Advances in Civil Engineering 24th-25th January 2014

[4] Minas K. Minoglou, George D. Hatzigeorgiou, George A. Papagiannopoulos "Heuristic optimization of cylindrical thin-walled steel tanks under seismic loads", Elsevier, Thin-Walled Structures 64 (2013) 50–59

[5] James I. Chang, Cheng-Chung Lin "A study of storage tank accidents", Elsevier, Journal of Loss Prevention in the Process Industries 19 (2006) 51–59

[6] B. S. Thakkar, S. A. Thakkar, "Design Of Pressure Vessel Using ASME Code, Section VIII, Division 1", IJAERS Vol. I, Issue II, March 2012/228-234

[7] Kalyan Kumar Mandal, Damodar Maity, "Pressure Based Eulerian Approach for Investigation of Sloshing inRectangular Water Tank", Elsevier, 12th International Conference on Vibration Problems, ICOVP 2015

[8] Wing Kam Liu, M. ASCE and Dennis Lam, "Nonlinear Analysis of Liquid-Filled Tank", ASCE, New York University, November 2015

[9] W. Maresca, Jr., Maria P. MacArthur, Angela Regalia, James W. Starr, Christopher P. Wilson, Robert M. Smedfjeld, "Pressure and Temperature Fluctuations in Underground Storage Tank Pipelines Containing Gasoline", Elsevier, Oil & Chemical Pollution 7 (1990) 29-56

[10] Paul F. Hudak, David J. Wachal, Bruce A. Hunter, "Managing subsurface property hazards: reactive soils and underground storage tanks", Elsevier, Urban Water 1 (1999) 237-241, April 2000

[11] David V. Hutton, Fundamental of Finite Element Analysis, McGraw Hill, 1 Edition, 2004