International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Issue 5, Volume 2 (May 2015) www.ijirae.com

STUDY ON SUBSEA STORAGE TANKS REINSTATEMENT OF STEEL WITH COMPOSITIES

SURESH KUMAR T
II YEAR ME STRUCTURAL ENGINEERING

CHAPTER 1
INTRODUCTION

1.1 GENERAL

Large quantity of oil is obtained from offshore wells, many of which are many miles from the nearest land. While the oil from such wells can be transported by the pipelines to shore for storage, it is generally considered more suitable and economical for the oil to be stored close to the producing well until the amount accumulated is adequate for a tanker to remove the oil and transport to its suitable destination. Usually these subsea storage tanks are a combination of various shell forms with non-negative geometric conditions such as cylindrical shell, conical shell, spherical shells etc… The material used for these subsea storage tanks are usually steel, concrete, composites or a combination. Shell structures are widely used in all industrial applications; especially those related to automobile, marine, nuclear, civil, aerospace and petrochemical engineering.

Steel, concrete or combinations of both were usually used materials for the construction of subsea storage tanks. Steel is mainly used for the construction of storage tank, because steel has a large load carrying capacity and due to its structural integrity. Steel plates can be obtained in any shape and size and they are usually economical. The main disadvantage of the steel structure is its weight and the fact that steel is easily subjected to corrosion. This is where composites come into picture. Advanced fiber reinforced composites are used for construction because of their high stiffness, modulus to weight ratio, strength to weight ratio and high corrosion resistant. These material properties help them to meet the design requirements flexibly. This is the main advantage of using composite shells for subsea. Use of composites for subsea shells allows economic exploitation of doubly curved shells, spheres, spheroids, ellipsoids etc… Thin walled composite materials offer many advantages in construction. As well as saving weight, they are cheaper to manufacture and also to maintain, because they need less inspection over their life time than equivalent steel structures. Further composites do not react with many aggressive chemicals, so they are useful to make reaction vessels for the chemical industry.

1.2 OBJECTIVES

The Objectives of this study are the following:

1. Static analysis of a subsea oil storage tank made of steel.
2. Reinstate the steel part with suitable composite.
3. Static analysis of the reinstated oil storage tank.

1.3 METHODOLOGY

Static analysis is done using finite element software ANSYS. The model for the Steel structure and the Reinstate composite structure will be created and the properties will be evaluated for the different loading conditions. The results will be tabulated and studied for the properties.

CHAPTER 2
FINITE ELEMENT ANALYSIS

FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system. FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such astropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas. Finite element analysis (FEA) has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the

© 2014-15, IJIRAE- All Rights Reserved

method is so important that even introductory treatments of Mechanics of Materials – such as these modules – should outline its principal features. In spite of the great power of FEA, the disadvantages of computer solutions must be kept in mind when using this and similar methods: they do not necessarily reveal how the stresses are influenced by important problem variables such as materials properties and geometrical features, and errors in input data can produce wildly incorrect results that may be overlooked by the analyst.

Perhaps the most important function of theoretical modeling is that of sharpening the designer's intuition; users of finite element codes should plan their strategy toward this end, supplementing the computer simulation with as much closed-form and experimental analysis as possible. Finite element codes are less complicated than many of the word processing and spreadsheet packages found on modern microcomputers. Nevertheless, they are complex enough that most users do not find it effective to program their own code. A number of prewritten commercial codes are available, representing a broad price range and compatible with machines from microcomputers to supercomputers. However, users with specialized needs should not necessarily shy away from code development, and may find the code sources available in such texts as that by Zienkiewicz to be a useful starting point. Most finite element software is written in Fortran, but some newer codes such as felt are in C or other more modern programming languages.In practice, a finite element analysis usually consists of three principal steps:

1. **Preprocessing:** The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete subregions, or "elements," connected at discrete points called "nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.
2. **Analysis:** The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations Kijuj = fi where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.
3. **Postprocessing**: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay colored contours representing stress levels on the model, showing a full-field picture similar to that of photoelastic or more experimental results. The operation of a specific code is usually detailed in the documentation accompanying the software, and vendors of the more expensive codes will often offer workshops or training sessions as well to help users learn the intricacies of code operation.

One problem users may have even after this training is that the code tends to be a "black box" whose inner workings are not understood. In this module we will outline the principles underlying most current finite element stress analysis codes, limiting the discussion to linear elastic analysis for now. Understanding this theory helps dissipate the black-box syndrome, and also serves to summarize the analytical foundations of solid mechanics.

CHAPTER 3
STRUCTURAL BEHAVIOR OF SUBSEA SHELLS

3.1 LOADS

Subsea structures are usually composed of shell components because load carrying capacity of shell structures is more when compared to other geometry. Subsea shells are usually subjected to loads both static and dynamic in nature. The most common types of loadings are the axial compression, bending, torsion and lateral pressure. These loadings not only challenge the strength of the structure but can also cause deformation of unacceptably large amplitude and could lead to loss of stability and collapse of the whole structure. Distributed loads due to internal pressure in storage tanks, pressure vessels or silos or to external pressure from wind, marine currents and hydrostatic pressures are very well resisted by the in-plane behavior of shells. On the other hand, concentrated loads introduce significant local bending stresses which have to be carefully considered in design. Such loads can be due to vessel supports or in some cases, due to abnormal impact loads in containment buildings of nuclear power plants, for example, codes of practice usually require the possibility of missile impact or even sometimes airplane crashes to be considered in the design. In these cases, the dynamic nature of the load increases the danger of concentrated effects.

© 2014-15, IJIRAE- All Rights Reserved

3.2 BEHAVIOR OF STEEL OR CONCRETE SHELLS

Curved surface structures are known as shells. Examples of shells include pressure vessels, airplane wings, pipes, the exterior of rockets, missiles, automobile tires, incandescent lamps, caps, roof domes, factory or car sheds and a variety of containers. Each of these has walls that are curved. The plane bisecting the shell thickness is called the midsurface. To describe the shape of a shell, we need only know the geometry of the midsurface and the thickness of the shell at each point. Shells of technical significance are often defined as thin when the ratio of thickness t to radius curvature r is equal to or less than 1/20. For thin shells of practical importance, this ratio may be 1/1000 or smaller. Shell structures by virtue of their shell geometry carry the applied loads primarily by direct stresses lying in their plane accompanied by a little or no bending. External hydrostatic pressure induces compressive stress resultants in the shells and may cause buckling at a pressure, much lower than the axisymmetric yield. Subsequently analytical investigation on buckling of such shell forms is the major problem to be addressed. The introduction of stiffeners considerably increases the buckling strength of the shell and is a satisfactory solution for increasing the strength of the shell.

Stress and displacement values for different types of shells are explained below:

Cylindrical shell:

Hoop stress; $\sigma_a = \frac{Pr}{2t}$ (3.2.1)

Meridional stress; $\sigma_h = \frac{Pr}{t}$ (3.2.2)

Deflection; $\delta = \frac{pr^2}{2Et}(2-\nu)$ (3.2.3)

Spherical shell:

Hoop stress; $\sigma_a = \frac{Pr}{4t}$ (3.2.4)

Meridional stress; $\sigma_h = \frac{Pr}{4t}$ (3.2.5)

Deflection; $\delta = \frac{pr^2}{2Et}(1-\nu)$ (3.2.6)

3.3 BEHAVIOR OF COMPOSITE SHELLS

A composite material is defined as a material system which consists of a mixture or a combination of two or more distinctly differing materials which are insoluble in each other and differ in form or chemical composition. Fiber reinforced composite materials consist of fibers of significant strength and stiffness embedded in a matrix with distinct boundaries between them. Both fibers and matrix maintain their physical and chemical identities, yet their combination performs a function which cannot be done by each constituent acting singly. Fibers of fiber reinforced plastic (FRP) may be short or continuous. It appears obvious that FRP having continuous fibers is indeed more efficient. The major constituents of a fiber reinforced composite material are reinforcing fiber, matrix, coupling agent, coatings and fillers. Fibers are the principal load carrying members while the matrix which surrounds it, keeps them in proper location and correct orientation. Matrix acts as the medium by which the load is transferred through the fibers by means of shear stress.

3.4 RECOMMENDED TYPES OF ANALYSIS

Types of analysis recommended for subsea shells are linear static analysis, linear buckling analysis and nonlinear analysis

3.4.1 LINEAR STATIC ANALYSIS

Linear static analysis is the strength analysis in which the principle of superposition is valid. It is based on the small deflection theory where stress strain relations and strain displacement relations are linear. In this method of analysis the change in geometry of the structure is not taken into account while deriving the equilibrium equations. The linear static analysis of the shell structures can be performed by solving the general finite element equilibrium equations, consisting of linear elastic stiffness matrix and load vector. Deformation pattern and stress resultants can be calculated. Elastic analysis always gives the change of stress or change of deflection due to a load or pressure, provided yielding does not occur. Elastic analysis therefore only gives the changes of stress which are additional to any stress in the structure at zero load. It is important when fatigue is a probable mode of failure and in calculating deflections and the stiffness of the structure.

3.4.2 LINEAR BUCKLING ANALYSIS

Buckling phenomenon is the major failure mode associated with thin walled cylindrical structures subjected to external pressure. The structure can suffer instability at a pressure, which may be only a small fraction to cause material failure. The buckling phenomenon associated with thin walled circular cylindrical shell subjected to uniform external pressure can be explained using the load deflection curve shown in fig. (Rajagopalan, 1993).

© 2014-15, IJIRAE- All Rights Reserved

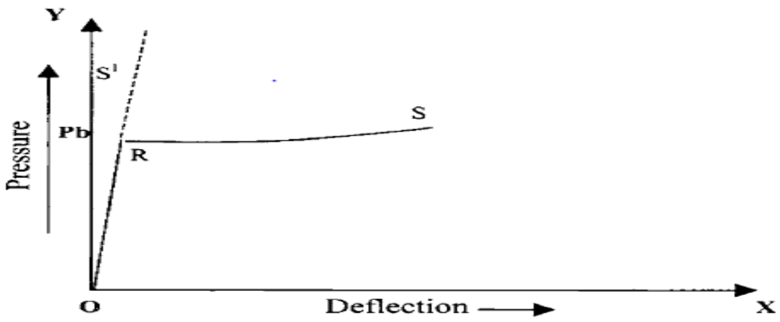

Fig: 3.1 Load deflection curve

The first regime OR, called the prebuckling state, determines the axisymmetric state of stress due to axisymmetric pressure load on the perfect cylinder. The prebuckling path is linear. The second regime RS, called the buckling stage and the load deflection curve for a perfectly circular cylinder subjected to uniform external pressure splits into two at the point R. At this point the load deflection curve can be either RS or RS1 and the pressure Pb is called bifurcation buckling pressure.

3.4.3 NONLINEAR ANALYSIS

In structural mechanics a problem is nonlinear if the stiffness matrix or load vector depends on displacements. The cause of nonlinearity may be material or geometric. The material nonlinearity may be due to nonlinear stress-strain relations and geometric nonlinearity may be due to nonlinear kinematic relations i.e. nonlinear strain-displacement relations (large displacements) and large strains. The geometric nonlinearity in which the nonlinear effect arising from nonlinear strain displacement relations and nonlinearity due to follower force effect of hydrostatic pressure are to be taken into consideration for stiffened cylindrical shell subjected to hydrostatic pressure. These two are smooth non linearites and incremental iterative procedure can effectively be used as solution strategy.

CHAPTER 4
FINITE ELEMENT ANALYSIS OF OIL STORAGE STRUCTURE

4.1 DESCRIPTION OF GEOMETRY OF THE STRUCTURE

The structure considered here is a subsea storage tank which is used to store crude oil. It has a base diameter of 83m and a height of 63m. The structure is fully immersed in water. The geometry of the structure is such that it is a combination of different types of shell structures. It contains a top cylindrical funnel, a conical transition, spherical storage tank and a bottom cylindrical tank. Except the bottom cylindrical tank, all the other parts are made of steel and the bottom cylindrical tank is made of concrete. The structure has varying thickness. It is a gravity type, bottom less structure which works on water displacement principle. The tank will be either full of oil or water or a combination of both. This structure is presently in use in the Persian sea. A figure showing the dimensions and thickness of different parts is given below.

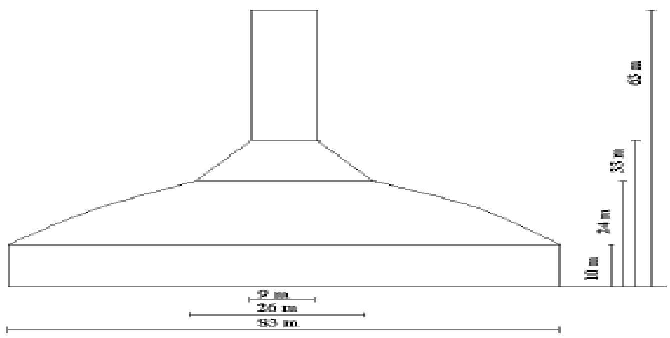

Fig: 4.1 Geometry of the considered storage tank

4.2 DESCRIPTION OF FINITE ELEMENT SOFTWARE

The finite element software used here is ANSYS. ANSYS is a finite element analysis (FEA) software package. It uses a preprocessor software engine to create geometry. Then it uses a solution routine to apply loads to the meshed geometry.

© 2014-15, IJIRAE- All Rights Reserved

Finally it outputs desired results in post-processing. The ANSYS program has many finite element analysis capabilities ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The primary objective of a finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. The ANSYS program provides a variety of ways for the application of loads. With the help of load step options, we can control how the loads are actually used during solution. The initial state capability of ANSYS allows defining a nontrivial state from which to start an analysis. The term initial state refers to the state of a structure at the start of an analysis. In ANSYS we can specify an initial stress, strain, or plasticity state for a structure. In the solution phase of an analysis, the computer takes over and solves the simultaneous set of equations that the finite element method generates. The ANSYS program writes the results to the data base as well as to the results file. The ANSYS program uses a data base to store all the data that is defined during an analysis.

4.3 MODELING OF THE STRUCTURE

4.3.1DESCRIPTION OF ELEMENT

The element used for modeling the considered structure is SHELL 181. The SHELL 181 element is suitable for analyzing thick to moderately thick shell structures. The element has four nodes with six degrees of freedom at each node; translations in the x, y and z axis and rotations along the x, y, and z axis. The degenerate triangular option should only be used as filler elements in mesh generation.SHELL 181 is well suited for linear, large rotation, and/or large strain nonlinear applications. In the element domain, both full and reduced integration schemes are supported. The element accounts for follower effects of distributed pressures. SHELL 181 is used for modeling the composite structure also because SHELL 181 may be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.

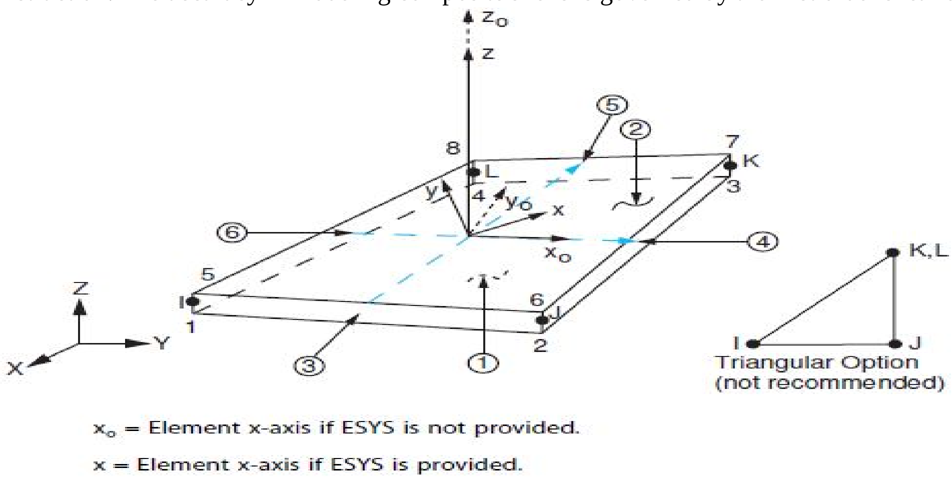

Fig: 4.2 SHELL 181

4.3.2 GEOMETRY

The structure considered here is an axisymmetric structure. Due to the symmetry of the structure, only one quarter of the structure need to be modeled.

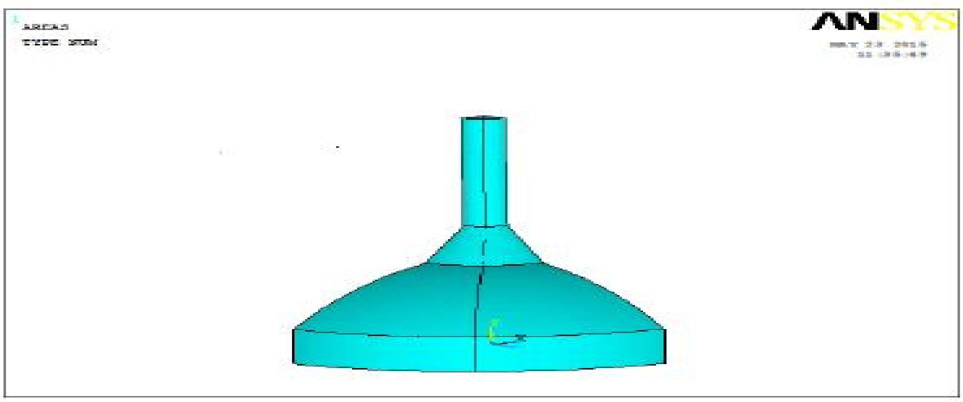

Fig: 4.3 Structure modeled in ANSYS

© 2014-15, IJIRAE- All Rights Reserved

4.3.3 BOUNDARY CONDITIONS

The structure is such that it rests on the ground due to the effect of gravity. Therefore the boundary condition applied at the base is fixed boundary condition. The top portion is considered as free, as the two sides of the quarter model is set as continuous in x direction.

4.3.4 MESHING

Meshing is an integral part of the computer-aided engineering simulation process. The mesh influences the accuracy, convergence and speed of the solution. The goal of meshing in ANSYS is to provide robust, easy to use meshing tools that will simplify the mesh generation process. These tools have the benefit of being highly automated along with having a moderate to high degree of user control. The structure considered is meshed using quadrilateral element for better accurate result.

4.3.5 LOAD APPLICATION

The next step after meshing is application of loads. In this case the load acting is external or internal water/oil pressure acting on the surface of the structure. In the considered structure, since the diameter is very much greater than the height of the structure pressure can be considered to be uniformly acting. Here three load cases are considered i.e.

- Empty tank submerged condition
- Submerged tank full of water
- Submerged tank full of oil

The pressure acting for different load cases are

1. Empty tank submerged condition = 6344.15kN/m^2
2. Submerged tank full of water = 3915.09kN/m^2
3. Submerged tank full of oil = 4215.48kN/m^2

4.3.6 ANALYSIS OF STEEL STRUCTURE

The steel structure modeled above is analyzed for the considered load conditions. Linear elastic analysis and linear buckling analysis were done.

4.3.6.1 LINEAR ELASTIC ANALYSIS

Results obtained in linear elastic analysis are shown below:

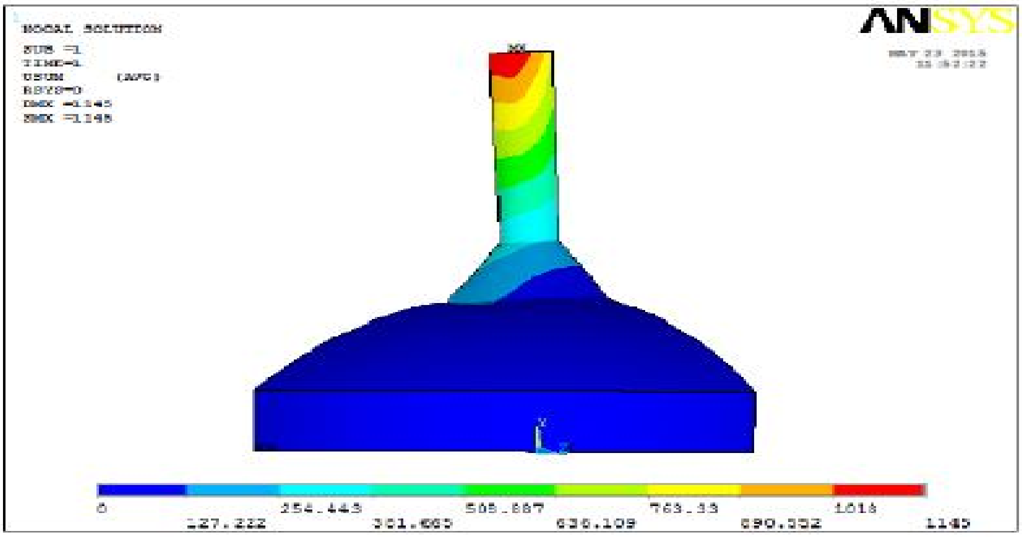

Fig: 4.4: Deflected shape showing max: deflection for empty steel tank submerged

© 2014-15, IJIRAE- All Rights Reserved

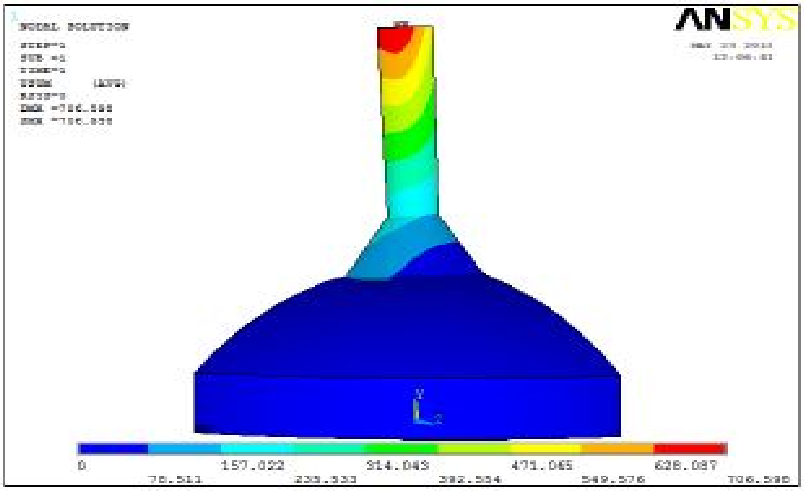

Fig: 4.5: Deflected shape showing max: deflection for submerged steel tank full of water

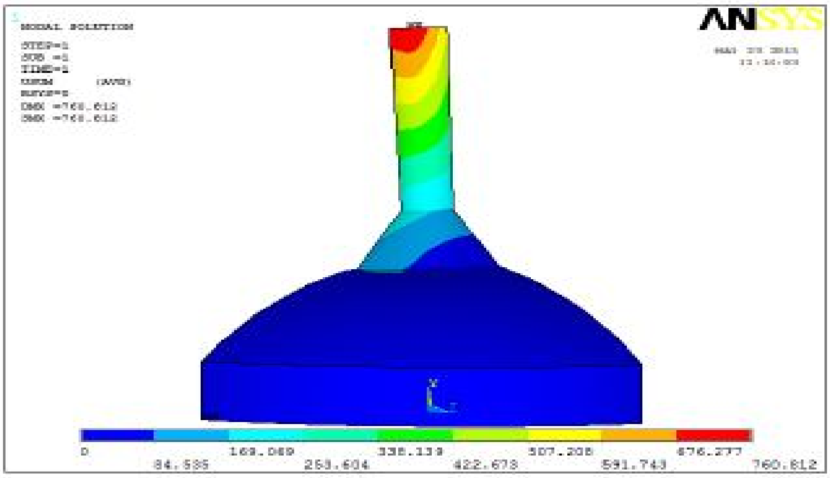

Fig: 4.6: Deflected shape showing max: deflection for submerged steel tank full of oil

4.3.6.2 LINEAR BUCKLING ANALYSIS

Linear buckling analysis for all the load conditions was done. It was done for 3 mode shapes, the results obtained are:

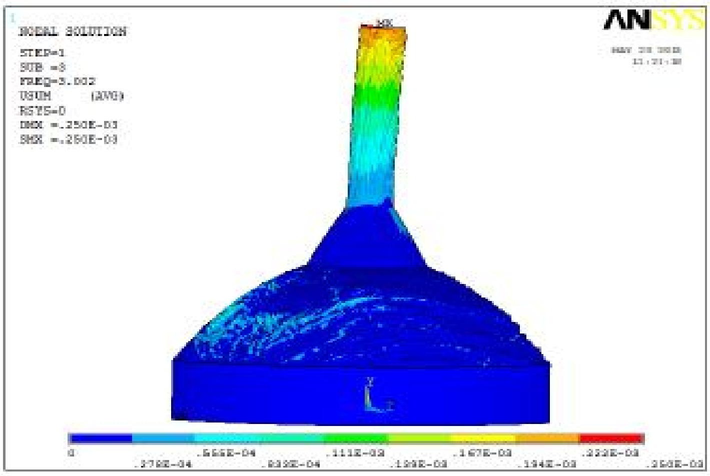

Fig: 4.7 Third Mode buckling for empty steel tank submerged

© 2014-15, IJIRAE- All Rights Reserved

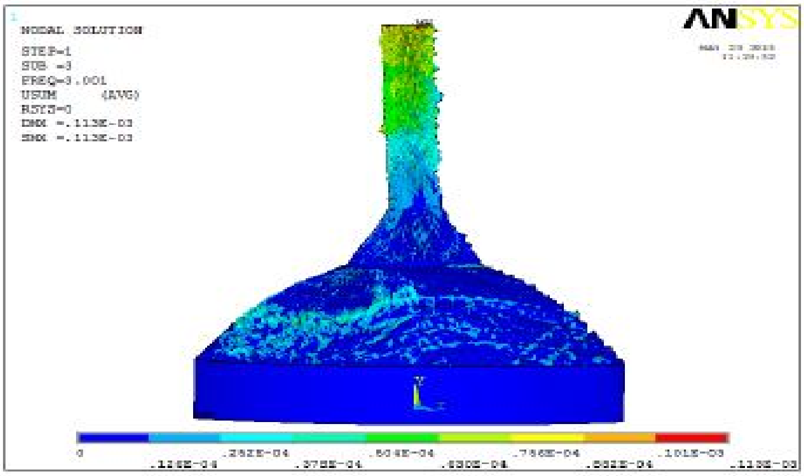

Fig: 4.8 Third Mode buckling for submerged steel tank full of water of oiL

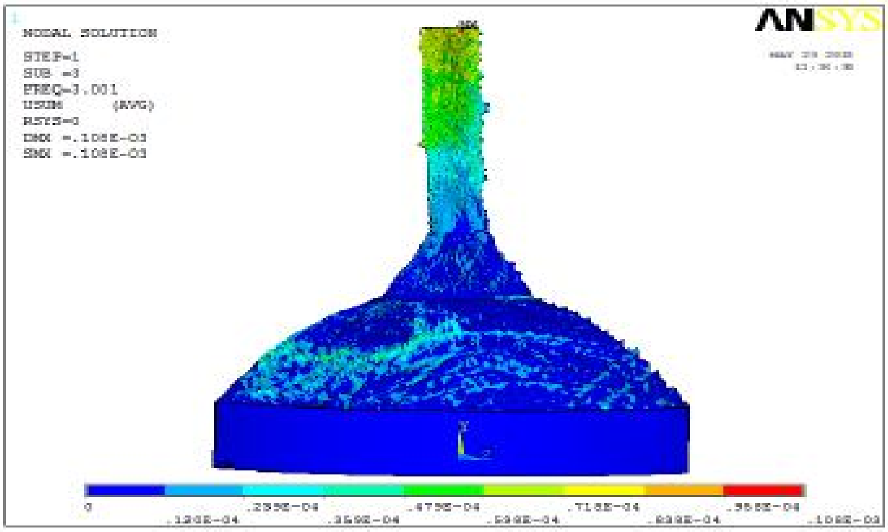

Fig: 4.9 Third Mode buckling of submerged steel tank full of oil

4.4 DESCRIPTION OF REINSTATE

Here carbon FRP is used to reinstate steel. CFRP is an extremely strong and light fiber reinforced polymer which contains carbon fibers. The polymer is most often epoxy, but other polymers, such as polyester, vinyl ester or nylon, are sometimes used. The composite may contain other fibers, such as aramid e.g. Kevlar, Twaron, aluminium, or glass fibers, as well as carbon fiber. The compound is also used in sailboats, rowing shells, modern bicycles, and motorcycles, where its high strength-to-weight ratio and very good rigidity is of importance. Improved manufacturing techniques are reducing the costs and time to manufacture, making it increasingly common in small consumer goods as well. Carbon-fiber-reinforced polymers are composite materials. In this case the composite consists of two parts; a matrix and reinforcement. In CFRP the reinforcement is carbon fiber, which provides the strength. The matrix is usually a polymer resin, such as epoxy, to bind the reinforcements together. Because CFRP consists of two distinct elements, the material properties depend on these two elements. The reinforcement will give the CFRP its strength and rigidity; measured by Stress (mechanics) and Elastic modulus respectively. Unlike isotropic materials like steel and aluminum, CFRP has directional strength properties. The properties of CFRP depend on the layouts of the carbon fiber and the proportion of the carbon fibers relative to the polymer. Here in this problem CFRP is reinstated steel in the storage tank, for the same thickness as that of steel. Carbon fibers have a very high tensile strength and elastic modulus. The elastic modulus of "high modulus" carbon fiber is similar to steel. Usually young's modulus of carbon fiber reinforced polymer is 120 – 580 GPa. In this case young's modulus is taken as 350 GPa, and poisons ratio is taken as .074

© 2014-15, IJIRAE- All Rights Reserved

International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Issue 5, Volume 2 (May 2015) www.ijirae.com

4.5 ANALYSIS OF COMPOSITE REINSTATE STRUCTURE

The reinstate composite storage tank is analyzed for the considered load conditions. Linear elastic analysis and linear buckling analysis were done.

4.5.1 LINEAR ELASTIC ANALYSIS

Linear elastic analysis was done and the obtained results are shown below.

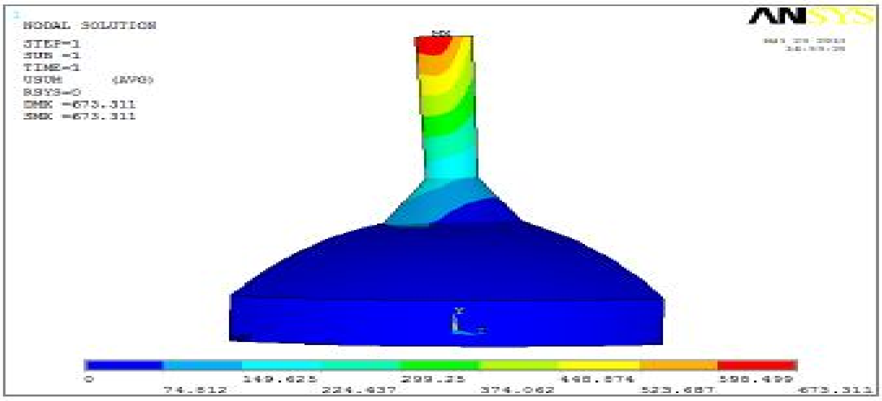

Fig: 4.10: Deflected shape showing max: deflection for empty composite tank submerged

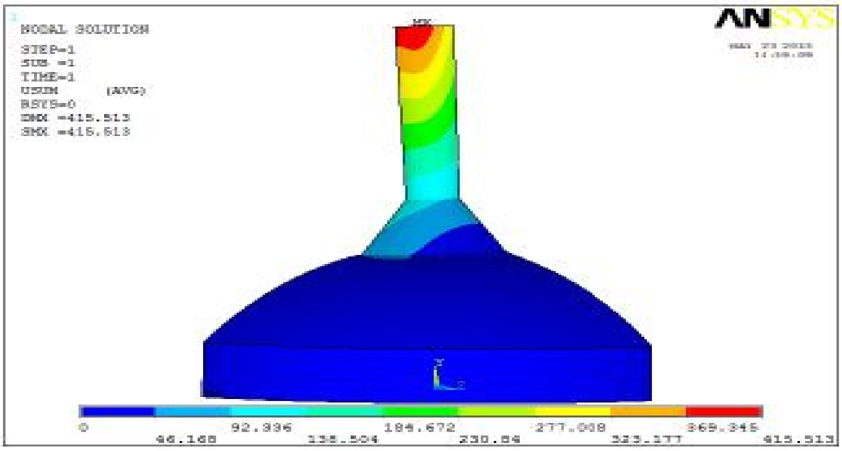

Fig: 4.11: Deflected shape showing max: deflection for submerged composite tank full of water

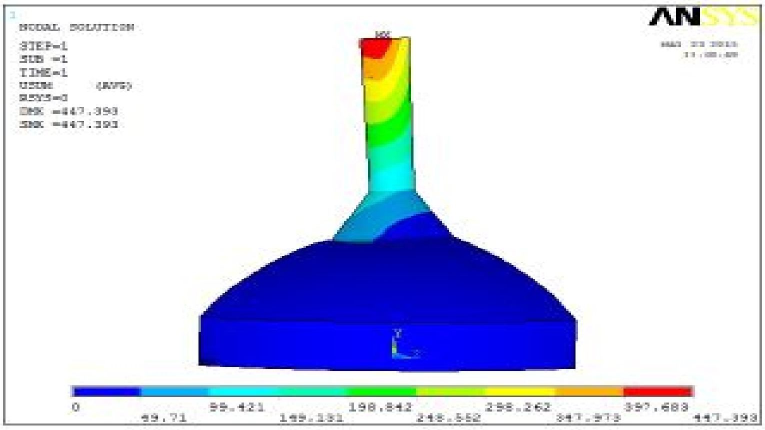

Fig: 4.12: Deflected shape showing max: deflection for submerged composite tank full of oil

4.5.2 LINEAR BUCKLING ANALYSIS

Linear buckling analysis was carried out and the obtained results are shown below,

© 2014-15, IJIRAE- All Rights Reserved

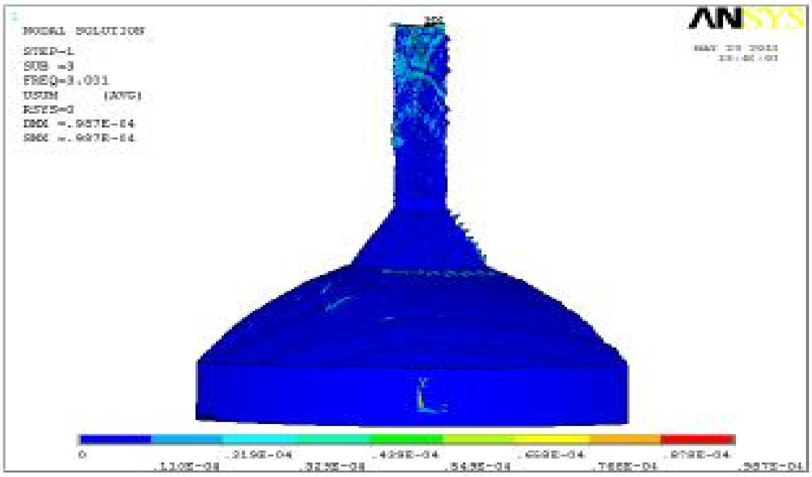

Fig: 4.13 Third Mode buckling of empty composite tank submerged

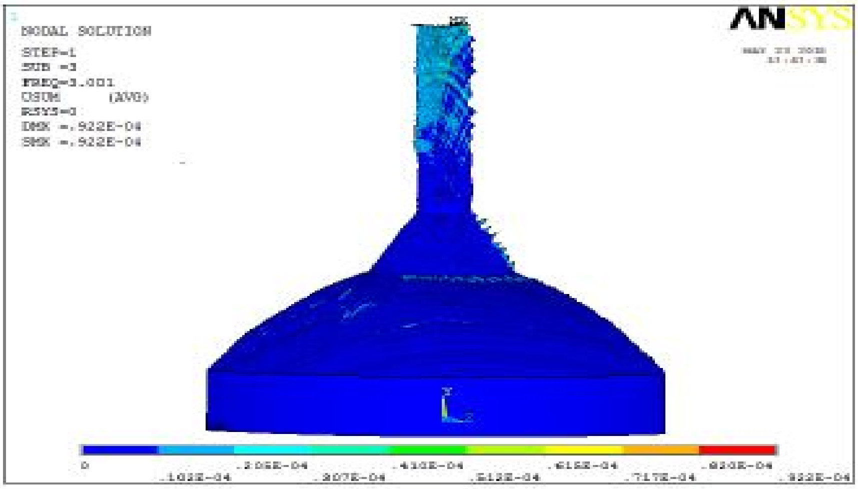

Fig: 4.14 Third Mode buckling of composite tank full of water

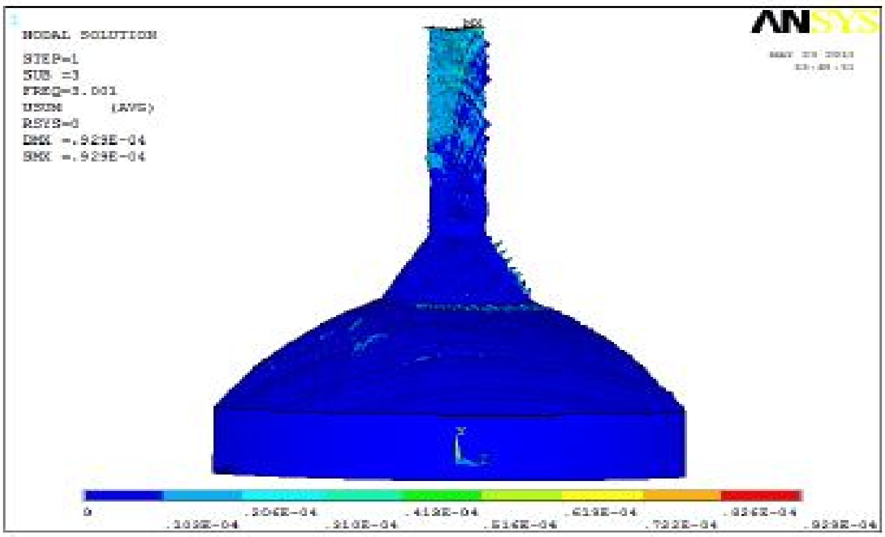

Fig: 4.15 Third Mode buckling of composite tank full of oil

International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Issue 5, Volume 2 (May 2015) www.ijirae.com

CHAPTER 5
CONCLUSION

5.1 RESULTS

Linear static analysis of the steel structure and the newly reinstated composite structure for three different load conditions were done using finite element software ANSYS, and results were obtained. The obtained results come under the permissible stress limits of steel and composites as per the standards of ASMI section II, for both the cases. It was observed that as compared to steel the deflection values of composite structure for the same load conditions were very less. Whereas there are no much difference in the stress values. Linear buckling analysis was also done for all the considered load conditions. Buckling factors were extracted for three modes. The obtained results are such that the buckling factor for steel is more than that compared to composites. Stress and deflection values and buckling factors for steel and composites for different load cases are given below.

Table 5.1 Stresses and deflection for empty tank submerged

	Steel	Composites
Deflection	1145mm	673.31mm
von Mises stress	41.44 x 10^3kN/m^2	39.78 x10^3kN/m^2
Meridional stress	-19.8 x 10^8kN/m^2	-20.7 x10^8kN/m^2
Hoop stress	-10.9 x 10^8kN/m^2	-10.3 x 10^8kN/m^2

Table 5.2 Stresses and deflection for submerged tank full of water

	Steel	Composites
Deflection	706.6mm	415.5mm
von Mises stress	25.6 x 10^3kN/m^2	24.55 x10^3kN/m^2
Meridional stress	-12.2 x 10^8kN/m^2	-12.8 x10^8kN/m^2
Hoop stress	-6.73 x 10^8kN/m^2	-6.37 x 10^8kN/m^2

Table 5.3 Stresses and deflection for submerged tank full of oil

	Steel	Composites
Deflection	760.8mm	447.4mm
von Mises stress	27.54 x 10^3kN/m^2	26.44 x10^3kN/m^2
Meridional stress	-13.2 x 10^8kN/m^2	-13.8 x10^8kN/m^2
Hoop stress	-7.24 x 10^8kN/m^2	-6.85 x 10^8kN/m^2

Table 5.4 Buckling factors for empty tank submerged

Mode No:	Steel	Composites
1	.148x10^{-3}	.621x10^{-4}
2	.114x10^{-3}	.416x10^{-4}
3	.25x10^{-3}	.987x10^{-4}

Table 5.5 Buckling factors for submerged tank full of water

Mode No:	Steel	Composites
1	.251x10^{-3}	.615x10^{-4}
2	.163x10^{-3}	.416x10^{-4}
3	.113x10^{-3}	.922x10^{-4}

Table 5.6 Buckling factors for submerged tank full of oil

Mode No:	Steel	Composites
1	.262x10^{-3}	.616x10^{-4}
2	.159x10^{-3}	.461x10^{-4}
3	.108x10^{-3}	.929x10^{-4}

5.2 CONCLUSION

- A reinstate using CFRP for the steel storage tank has been proposed.
- Deflections of the composite structure is found lower than that of steel, whereas the steel storage tank has a higher buckling load factor which indicates that the load carrying capacity of steel tank is greater than that of composite.
- There is a weight reduction of 21% for composite compared to steel

© 2014-15, IJIRAE- All Rights Reserved

CHAPTER 6
REFERENCE

[1]. Aditya, A.K. and Bandyopadhyay, J.N. (1989). Simplified bending analysis of doubly curved shells, *Computers & Structures,* 33(3): 781-784

[2]. Ambartusumyan SA. Theory of anisotropic shells. NASA-TT-F-118; 1964.

[3]. ANSYS tutorial

[4]. AsadiE, Wang, QatuMS. Static and vibration analyses of thick deep laminate cylindrical shells using 3D and various shear deformation theories. Composite Structures2012; 94:494–500.

[5]. Bhimaraddi, A. (1984). A higher order theory for free vibration analysis of circular cylindrical shells. *Int. J. Solids Structures*, 20(7): 623-630.

[6]. Bhimaraddi A, Carr AJ, Moss PJ. A shear deformable finite element for the analysis of general shells of revolution. Comput Struct 1989;31(3):299–308

[7]. Cho, M., Kim, K. and Kim, M. (1996). Efficient higher-order shell theory for laminated composites. *Composite Structures*, 34(2): 197-212.

[8]. Choi, C.K. (1984). A conoidal shell analysis by modified isoparametric element, *Computers & Structures,* 18(5): 921-924.

[9]. Flügge W. Stresses in shells. Springer-Verlag; 1960.

[10]. Ghosh, B. and Bandyopadhyay, J.N. (1989). Bending analysis of conoidal shells using curved quadratic isoparametric element, *Computers & Structures*, 33(4): 717-728.

[11]. Ghosh, B. and Bandyopadhyay, J.N. (1990). Approximate bending analysis of conoidal shells using the Galerkin method, *Computers & Structures,* 36(5): 801-805.

[12]. Gol'Denveizer AL. Theory of elastic thin shells. Pergamon Press; 1961.

[13]. G.R. Heppler, L. Wahl, Finite element analysis of free–free shells of revolution, Journal of Sound and Vibration 152 (2) (1992) 263–283.

[14]. Kalnins, Analysis of shells of revolution subjected to symmetrical and nonsymmetrical loads, Transactions of the ASME 31E Journal of Applied Mechanics 3 (1964) 467–476.

[15]. Kant, T. and Khare, R.K. (1997). A higher-order facet quadrilateral composite shell element, *Int. J. for Numerical Methods in Engng.*, 40: 4477-4499.

[16]. Kant T, Kommineni JR. Geometrically non-linear analysis of sandwich shells with a higher-order theory and C0 finite elements. Compos Struct 1994;27:403–18.

[17]. S. Klein, Vibrations of multi-layer shells of revolution under dynamic and impulsive loading, Shock and Vibration Bulletin 35 (1966) 486–494.

[18]. Kraus H. Thin elastic shells. New York: John Wiley & Sons; 1967

[19]. Leissa AW. Vibration of plates. NASA-SP-160; 1969.

[20]. Leissa AW. Vibration of shells. NASA-SP-288; 1973.

[21]. Mallikarjuna and Kant, T. (1992). A general fiber-reinforced composite shell element based on a refined shear deformation theory. *Computers & Structures*, 42(3): 381-388.

[22]. Markuš Š. The mechanics of vibrations of cylindrical shells. Elsevier; 1988.

[23]. Novozhilov VV. Thin shell theory. P. Noordhoff; 1964.

[24]. Pinto Correia IF, Barbosa JI, Mota Soares CM, Mota Soares CA. A finite element semi-analytical model for laminated axisymmetric shells: statics, dynamics and buckling. Comput Struct 2000;76(1-3):299–317.

[25]. Reddy JN, Liu CF. A higher order shear deformation theory of laminated elastic shells. Int J Eng Sci 1985;23(3):319–30

[26]. Shu, X. and Sun, L. (1994). An improved simple higher-order theory for laminated composite plates. *Computers & Structures*, 50(2): 231-236.

[27]. T.A. Smith, Analysis of axisymmetric shell structures under axisymmetric loading by the finite element method, 1966 US Army Missile Command Technical Report RS-TR-66-8, Redstone Arsenal, Alabama, 1966

© 2014-15, IJIRAE- All Rights Reserved