ISSN 2348 – 8034
Impact Factor- 3.155

GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES

DESIGN AND ANALYSIS OF FRB STORAGE TANK

Balasubramanian.M*[1] and Ahamed Fahad.J[2]
*[1,2]M.E.Engineering design, Department of Mechanical Engg, K.Ramakrishnan College Of Engineering, Trichy- 621112

ABSTRACT
In the first part of this paper, a simplified approach to design a Shell & Tube Heat Exchanger [STHE] for beverage and process industry application is presented. The design of STHE includes thermal design and mechanical design. The thermal design of STHE involves evaluation of required effective surface area (i.e. number of tubes) and finding out log mean temperature difference [LMTD]. Whereas, the mechanical design includes the design of main shell under internal & external pressure, tube design, baffles design gasket, etc. The design was carried out by referring ASME/TEMA standards, available at the company. The complete design, fabrication, testing and analysis work was carried out at Alfa Laval (India), Ltd., Pune-12. In the second part of this paper detail view of design optimization is presented by flow induced vibration analysis [FVA].

Keywords- TEMA, HTRI.

I. INTRODUCTION
Heat Exchangers are devices used to enhance or facilitate the flow of heat. Every living thing is equipped in some way or another with heat exchangers. They are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas processing, and sewage treatment. The design of STHE including thermodynamic and fluid dynamic design, cost estimation and optimization, represents a complex process containing an integrated whole of design rules and empirical knowledge of various fields.
The design of STHE involves a large number of geo-metric and operating variables as a part of the search for heat exchanger geometry that meets the heat duty requirement and a given set of design constrains. A STHE is the most common type of heat exchanger in oil re-fineries and other large chemical processes, and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large vessel) with a bundle of tubes inside it. One fluid runs through the tubes and the second runs over the tubes (through the shell) to transfer heat between the two fluids. A set of tubes is called a tube bundle which may be composed by several types of tubes e.g. plain, longitudinally finned, etc.

1.1 Introduction to Thin Composite Shells
The shell whose wall thickness is small compared to the radius of curvature and the corresponding radius of twist is known as thin shell. Plate and shell structures are used in a lightweight load bearing structural parts for various modern aerospace, offshore, nuclear, automotive, and civil engineering structures. These shells are subjected to compressive loads. In the case of air crafts, they are subjected to fluctuating flight loads, which also produce compressive components. These compressive loads cause buckling of the shell structure. The analysis of composite shell structures requires consideration of a variety of failure modes. Often analysis programs cannot predict all failure modes using a single analysis model, and consequently structural designers must use a variety of analysis tools. It is also common that for a given failure mode, it is difficult to obtain the same result using different programs. There is no need to argue that composite shells are important in modern technology.

1.2 Composite Materials
Composites are considered to be combinations of materials differing in composition or form on a macro scale. The constituents retain their identities in the composite i.e. they do not dissolve or otherwise merge completely into each other although they act in the idea of a composite material is not a new or recent one concert. In nature, one can find out many composite materials, for example wood is a fibrous natural composite (cellulose fibrous in lignin matrix). Bone is yet another example of natural composites. Our ancestors invented composite by mixing straw and clay to make bricks. Straw is fiber reinforcement and clay is the matrix. The first glass fiber reinforced polymer was developed in 1940. The origin of distinct discipline of composite materials started in 1960‟s.Extensive research has been done on composite material since 1965. One difference between laminated composites and traditional engineering materials is that a composite‟s response to loads is direction dependent. Monolithic metals and their

alloys can"t always meet the demands of today"s advanced technologies. The composite materials exhibit high specific strength and high specific modulus resulting in substantial reduction of weight of the components, thus improves efficiency, and results in energy savings. One of the main advantages of composite materials is the flexibility involved in getting the desired strength and stiffness in the direction required. Carbon fibers are very common in high-modulus and high-strength applications. It is well known that the measured strength of most materials is much smaller than their theoretical strength because of presence of imperfections or inherent flaws in the materials. Flaws in the form of cracks that lies perpendicular to the direction of load are particularly detrimental to the strength. It is found that non polymeric materials have higher strength along their lengths because of small cross sectional fibers, the flaws are minimized. In case of polymeric materials, orientation of the molecular structure is responsible for high strength and stiffness. Fibers because of their small-sectional dimensions are not directly usable in engineering applications. They are embedded in matrix materials to form fibrous composites. The matrix serves to bind the fibers together; transfer loads to the fibers and protects them against environmental attack and damage due to handling. The shell is multi-layered fibrous composite shell. Each layer or lamina is a single-layer composite and thus orientation is varied according to design. Each layer is vary thin (thickness 0.4 mm to 0.7mm) and cannot be directly used. Several identical or different layers are bonded together to form a multi-layered composite shell. Each layer may be differing from another layer in (a) Relative volumes of the constituent"s materials (b) Form of the reinforcement. (c) Orientation of fibers with respect to common reference axes. Thus the directional properties of the individual layers may be quite different from each other.

composites is called the hot pressing method. Glass fibers in continuous tow are passed through slurry consisting of powdered matrix material, Solvent such as alcohol, and an organic binder as shown in fig 1.2. The tow is then wound on a drum and dried to form Prepreg tapes. The prepreg tapes can now be stacked to make a required laminate; heating at about 932°F (500°C) burns out the binder. Hot pressing at high temperatures of about 1832°F (1000°C) and pressures of 7 to 14Mpa. Design analysis of any composite structural element would require a complete knowledge of properties of individual layers. Each layer is a continuous angle-ply composite laminate consists of parallel fibers embedded in a matrix. Several unidirectional layers can be stacked in a specified sequence of orientation to fabricate a laminate that will meet design strength and stiffness requirements.

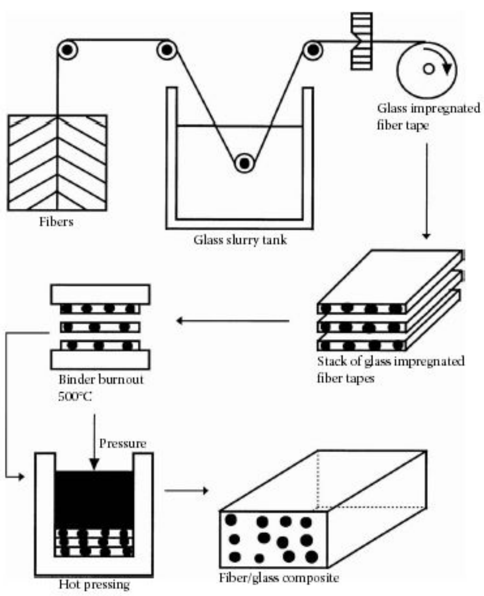


Fig. 1.2.Schematic Matrix Composite Laminates Manufacturing

Each layer of a unidirectional composite may be referred to as simple a layer ply or lamina. One of the most important factors for determining the properties of composites is relative proportions of the matrix and reinforced materials. The relative proportionate can be given as the weight fractions or by one of the experimental methods after fabrication. The volume fractions are exclusively used in the theoretical analysis of composite materials. Most

ISSN 2348 – 8034
Impact Factor- 3.155

manmade composite materials made from two materials, these are a reinforced material called fiber and a base material called matrix. For example in concrete columns the concrete is base material which is called matrix and the iron rods come under fibers for reinforcement.Their existing three types of composites.

1. **Fibrous Composites:** It consists of fibers of one material in a matrix material of another material.
2. **Particulate Composites:** These are composed of particles of one material in a matrix of another material.
3. **Laminate Composites:** These are made of layers in which fibers and matrix are made of different materials, including the composites.

The purpose of matrix is to transfer loads and protect them against environmental attack and damage due to handling. Based upon the properties required, the matrix and fiber materials are selected.

1.2.1. Fiber

Fibers are principal constituents in fiber reinforced composite material. They occupy the large volume fraction in a composite laminate and share major portion of load acting on a composite.
Glass is the most common fiber used in polymer matrix composites.
Carbon fibers are very common in high-modulus and high-strength applications. The advantages of carbon fibers include high specific strength and modulus, low coefficient of thermal expansion and high fatigue strength.

Aramid fiber is an aromatic organic compound made of carbon, hydrogen, oxygen and nitrogen. Its advantages are low density, high tensile strength, low cost and high impact resistance.

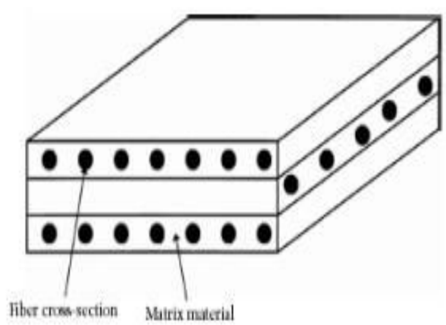


Fig1.3.Typical Laminate made of three laminate

II. EXISTING INDUSTRIAL SCENARIO

In industries, heat exchangers are used in industrial process to recover heat between two process fluids. Shell-and-tube heat exchangers are the most widely used heat exchangers in process industries because of their relatively simple manufacturing and their adaptability to different operating conditions. But nowadays numbers of industries are searching for effective and less time consuming alternatives of designing of shell-and-tube heat exchangers. As per literature and industrial survey it is observed that there is need of effective design options for STHE. This section explains the details of existing industrial scenario of design of STHE.

2.1. Part A-Thermal Design

The thermal design of STHE includes:
1) Consideration of process fluids in both shell and tube side;
2) Selection of required temperature specifications;
3) Limiting the shell and tube side pressure drop;
4) Setting shell and tube side velocity limits;
5) Finding heat transfer area including fouling factor.

2.2. Part B-Mechanical Design

The mechanical design of STHE includes:

1) Selection of TEMA layout—based on thermal de- sign;
2) Selection of tube parameters such as size, thickness, layout, pitch, material;
3) Limiting the upper and lower design on tube length;
4) Selection of shell side parameters such as material, baffle spacing, and clearances;
5) Thermal conductivity of tube material;
6) Setting upper and lower design limits on shell diameter and baffle spacing.

As per literature and industrial survey at [A1 & A2] the design is carried out using in-house developed software for design and drafting. This dedicated software enables qualified engineers to accomplish complex design calculations complying strictly with the requisite international codes and standards. The software also generates fabrication drawings to scale and 3-D images of the Exchanger thereby giving warning of any foul up/mismatch in nozzles, RF-Pads and in the dimensions of various components. Also an experienced team of design engineers undertakes thermal and mechanical design of complex heat exchangers and generate fabrication drawings to scale along with weights and estimates based on customer's specifications. These designs are optimized to arrive at an optimal size. After carrying out the design, the final output is in an AutoCAD drawing format (DWG) or DWF (Web format).

In this proposed work design, development & testing of STHE is carried out. Along with the parameter considered as per [A1 & A2], the software generated design was cross checked with manual design. Also vibration analysis is performed to optimize unsupported span of tube by using HTRI software.

The paper is organized as; detailed overview on work carried out by researchers is presented in Section 1 and Section 2 presents the existing industrial scenario of STHE design, Section 3 states the current problem definition & objective, detailed design (thermal and mechanical design) and details of STHE are given in Section 4. Section 5 explores manufacturing of STHE, while Section 6 describes hydraulic testing of STHE and concluding remark is given in Section 7.

III. PROBLEM DEFINITION & OBJECTIVE

The problem presented in this paper is to design & develop a STHE, conforming to the TEMA/ASME [1-2] Standards, based on following Input Data:

1) Inlet & Outlet Temperatures of fluids on Shell & Tube Side,
2) Tube length = 10,000 mm,
3) Tube OD = 38.1 mm,
4) Shell OD = 1350 mm.

As per the requirement the objective of the preset work is to perform thermal and mechanical design of STHE using TEMA/ASME standards to reduce time.

IV. DESIGN OF STHE

The design of STHE involves a large number of geometric and operating variables as a part of the search for an exchanger geometry that meets the heat duty requirement and a given set of design constrains. Usually a reference geometric configuration of the equipment is chosen at first and an allowable pressure drop value is fixed. Then, the values of the design variables are defined based on the design specifications and the assumption of several mechanical and thermodynamic parameters in order to have a satisfactory heat transfer coefficient leading to a suitable utilization of the heat exchange surface. The designer's choices are then verified based on iterative procedures involving many trials until a reasonable de- sign is obtained which meets design specifications with a satisfying compromise between pressure drops and thermal exchange performances [3-10].

The details of shell and tube heat exchanger under consideration are shown in **Table 1**.

The details of STHE are shown in **Table 1**. **Figure 1** shows the various major components of a typical STHE as listed below:

1) Connections
2) Tube Sheet
3) Gasket
4) Head/Dish End
5) Mounting/Support

ISSN 2348 – 8034
Impact Factor- 3.155

6) Baffles

7) Shell

8) Tube Bundle

V. MANUFACTURING OF STHE

In this section detail of manufacturing of STHE is ex- plained as shown in Figures 2-5

Table 1. STHE specification.

Parameter	Description
Size (Dia./length)	Ø1336/10,000 mm
Surface area (eff.)/unit	781.4 m. sq.
Shells/unit	1
Heat exchanged, (*Q*)	5064.9 KW
LMTD (Corrected)	9.15°C

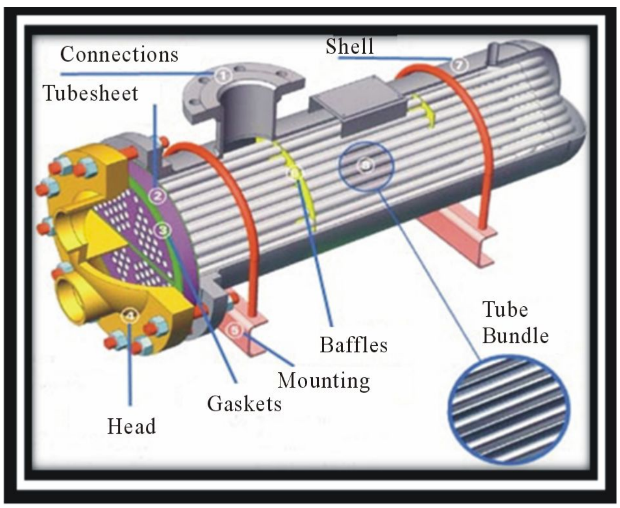


Figure 1. Major components of a typical STHE.

5.1. Fabrication of Main Shell and Channel Shell

The steps in fabrication of main shell and channel shell are stated as below:

Figure 2. Rolling of shell

ISSN 2348 – 8034
Impact Factor- 3.155

Figure 3. L-seam welding & DP test

Figure 4. Joining of shell

Figure 5. Tubes & Baffles assembly

- Material identification
- Marking
- Punching
- Cutting
- Grinding
- Edge preparation
- Rolling and L-seam fit up (**Figure 2**)
- L-seam welding (**Figures 3** and **4**)
- Back-chipping
- Die-penetrant (DP) Test (**Figure 3**)

5.2. Channel Shell to Dish End & Flange Fit-Up

- Material identification
- Edge preparation
- Flange Fit-up and Welding
- Channel Shell Dish End Fit-up and Welding

5.3. Tubes & Baffles Assembly

- Orientation marking
- Insertion of tie rods
- Insertion of spacers and baffles
- Insertion of tubes in the baffles
- Insertion of tube bundle assembly in the shell

5.4. Tube-Sheet Fit-Up

- Angular positioning of tube-sheet with respect to shell
- Welding of tube-sheet to shell
- Second tube-sheet fit-up

5.5. Tube to Tube-Plate Joint

- Length adjustment
- Tacking of the tubes
- Dye penetrant test
- Expansion of the tubes

5.6. Channels to Main Shell Fit-Up

- Gasket Positioning
- Bolting

VI. STATIC ANALYSIS AND BUCKLING ANALYSIS OF FEM

6.1 INTRODUCTION TO FINITE ELEMENT METHOD

The finite element method is a numerical procedure for analyzing structures and continua. Usually problem addressed is too complicated to solve satisfactorily by classical analytical methods. The finite element procedure develops many simultaneous algebraic equations, which are generated and solved on a digital computer. The results obtainable are accurate enough for engineering purposes at reasonable cost. In addition it is an efficient design tool by which designers can perform parametric design studies by considering various design cases(different shapes, materials, loads , etc.,), analyze them and choose the optimum design . Hence the method has increasingly gained popularity among both researchers and practitioners.

6.2 GENERAL DESCRIPTION OF FINITE ELEMENT ANALYSIS In the finite element method, the actual continuum or bodies of matter like solid, liquid or gas is represented as on assemblage of sub divisions called finite elements. These elements are considered to be interconnected at specified joints, which are called nodes or nodal points. The nodes usually lie on the element boundaries where adjacent elements are considered to be connected. Since the actual variation of the field variable (like displacement, stress, temperature, pressure and velocity) inside the continuum is not known, we assume that the variation of field variable inside a finite element can be approximated by a simple function. These approximating functions (also called interpolation models) are defined in terms of values at the nodes. When field equations (like equilibrium equations) for the whole continuum are written. The new unknown will be the nodal values of the field variable. By solving the field equations, which are generally

ISSN 2348 – 8034
Impact Factor- 3.155

in the form of matrix equations, the nodal values of the field variable will be known. Once these are known, the approximating function defines the field variable throughout the assemblage of elements.

6.3 BASIC STEPS OF FINITE ELEMENT ANALYSIS

There are three basic steps involved in this procedure

1) Pre Processor (Building the model (or) Modeling)

2) Solution (Applying loads and solving)

3) Post Processor (Reviewing the results)

Preprocessor (defining the problem) The problem is defined through mathematical modeling or analytical modeling. The formulation of the set of mathematical equations that models the physical problem within the scale and accuracy required by the application. The following steps are involved in preprocessor
• Define key point/lines/ areas/ volumes
• Define element type and material/ geometric properties
• Mesh lines/ areas / volumes are required.

Solution (assigning loads, constraints and solving) Here the loads (points load, uniformly distribute load, force or pressure), constraints (translational or rotational) are specified and finally solve the resulting set of equations. **Post Processor** The numerical results in terms of their mathematical and physical significance are interpreted. In this stage, further processing and viewing of the results cab be done such as
• List of nodal displacements
• Element forces and moments
• Deflection plots
• Stress contour diagrams

6.4 ELEMENTS USED FOR ANALYSIS

In Ansys (finite element analysis software) different types of elements are used depending on the problem nature and the type of analysis to be performed. The element chosen characterizes among other things, the degree of freedom set (displacements and/ or rotations, temperatures, etc.), the characteristic shape of the element (line, quadrilateral, brick, etc.), whether the element lies in 2-D space or 3 - D space, response of the system. The elements used in Ansys are classified as shown in table 6.1 below.

		ELEMENTS
Structural Point		MASS21
Structural Line	2-D	LINK1
	3-D	LINK8, LINK10, LINK11, LINK180
Structural Beam	2-D	BEAM3, BEAM23, BEAM54
	3-D	BEAM4, BEAM24, BEAM44, BEAM188, BEAM189
Structural Solid	2-D	PLANE2, PLANE25, PLANE42, PLANE82, PLANE83, PLANE145, PLANE146, PLANE182, PLANE183
	3-D	SOLID45, SOLID64, SOLID65, SOLID92, SOLID95, SOLID147, SOLID148, SOLID185, SOLID186, SOLID187
Structural Shell	2-D	SHELL51,SHELL61,SHELL208,SHELL209
	3-D	SHELL28, SHELL41, SHELL43, SHELL63, SHELL93, SHELL143, SHELL150, SHELL181
Structural Solid Shell	3-D	SOLSH 190
Structural pipe		PIPE16,PIPE17,PIPE18,PIPE20,PIPE59,PIPE60
Structural Interface		INTER192, INTER193, INTER194,INTER195
Structural Multipoint Constraint Elements		MPC184
Structural Layered Composite		**SOLID46**, SHELL91, SHELL99, SOLID191
Hyper Elastic Solid		HYPER56, HYPER58,HYPER74, HYPER84, HYPER86, HYPER 158
Thermal Line		LINK31, LINK32, LINK33, LINK34
Thermal Solid	2-D	PLANE35,PLANE55,PLANE75,PLANE77,PLANE78
	3-D	SOLID70,SOLID87,SOLID90
Thermal Shell		SHELL57,SHELL131,SHELL132

Table.6.1 List of Elements by Classification

6.5 ROLE OF FEM IN STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. The term structural or structure implies not only civil engineering structures such as ship hulls, aircraft bodies and machine housings as well as mechanical components such as pistons, machine parts and tools.

6.5.1 Types Of Structural Analysis

Different types of structural analysis are
• Static Analysis
• Model Analysis
• Harmonic Analysis
• Transient Analysis
• Spectrum Analysis
• Buckling Analysis
• Explicit Dynamic Analysis

6.6 STATIC ANALYSIS

A static analysis calculates the effects of the study loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can however, include steady inertia loads (such as gravity and rotational velocity), and time varying loads that can be approximated as static equivalent loads (such as static equivalent wind and seismic loads commonly defined in many building codes). Static analysis is used to determine the displacement, stresses strains and forces in structural components caused by loads that do not include significant inertia and damping effects. Steady loading and response conditions are assumed i.e. the loads and the structures response are assumed to vary slowly with respect to time. The kinds of loading that can be applied in a static analysis include:
• Entirely applied forces and pressures
• Steady static inertial forces (such as gravity (or) rotational velocity)

• Imposed (non-zero) displacements
• Temperatures (for thermal Strain)
• Fluences (for nuclear swelling)

A static analysis can be either linear or nonlinear. All types of nonlinearities are allowed- Like large deformations, plasticity, creep, stress, stiffening contact (gap) elements and hyper elastic elements.

6.6.1 Overview of Steps in Static Analysis

The procedure of static analysis involves three main steps
• Build the model
• Apply loads and obtain the solution
• Review the results

6.7 BUCKLING ANALYSIS

Buckling analysis is a technique used to determine buckling loads. Critical loads at which the structure becomes unstable and buckled.
• Build Model
• Obtain the static solution
• Obtain the eigen value buckling solution
• Expand the solution
• Review the results

Buckling analysis is a technique used to determine buckling analysis and element type, material properties; boundary conditions are same as for static analysis except the prestress effects are to be included.

Structural Analysis is probably the most common application of the finite element method provides a means of Optimizing the design with regard to static and buckling load, deformation , weight and material choice. Finite element analysis works to ensure accurate results by combining the designed geometric input with specific boundary condition information that is provided by process.

6.7.1 Modeling

In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represent the spatial volume and connectivity of the actual system. The ANSYS program offers you the following approaches to model generation: 3-D model was built for unstiffened and stiffened shell using CATIA Vr18 software. The modeled shell has the following properties: Cylinder diameter 670 mm Cylinder height 1030 Shell thickness 10 mm Stiffener Cross section 8 x 150mm The Thin walled composite cylinder is generated. a cylinder with stiffeners spaced periodically. It is used for calculating propagation constant in circumferential directions of the cylindrical shell subjected to circumferential mode.

6.7.2 MESHING

The Modeled cylinder was imported to ANSYS for meshing; The solid model was set element attributes, and established meshing controls, you can then turn the ANSYS program loose to generate the finite element mesh. By taking care to meet certain requirements, we can request a "mapped" mesh containing all quadrilaterals, all triangular, or all brick elements. Here the shell was meshed using quadrilateral elements. The mesh size used is 4mm for both shell and the stiffeners. The mesh size may be changed depending on the complexity of the problem. Also increases as the number of elements increases and it requires powered system to solve the problem. Mesh generation is one of the most critical aspects of engineering simulation.

ANSYS meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. The modeled cylinder has a radial symmetry of 45°. Initially one 45° sector was modeled and then the whole structure was generated using this primary sector. This can be treated as an axi-symmetric problem to reduce the complexity of the problem.

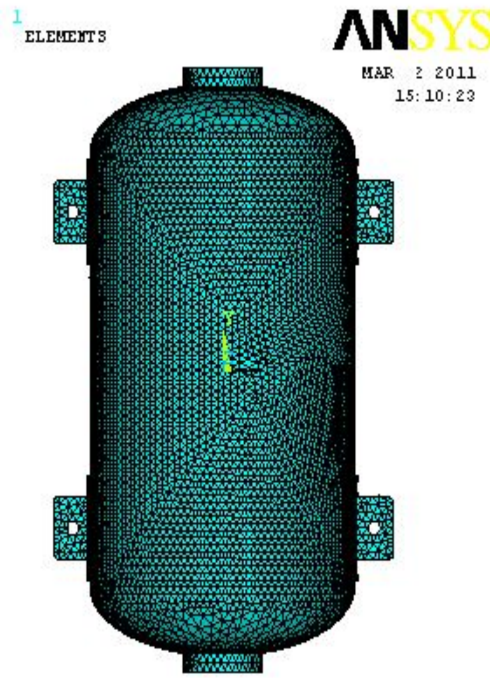


Fig.6.1. Meshed Model

a) Nodes:

Nodes are the basic building blocks of elements in a mesh. Nodes store elevation and other data set values. Nodes can also be used for building node strings and assigning boundary conditions. The density of mesh nodes helps determine the quality of solution data and can be important to model stability.

b) Mesh Element Types:

Elements are used to describe the area to be modeled. Elements are formed by joining nodes. The element types supported vary from model to model. Element types include:

- 1D elements
- Three-node line
- Triangular Elements
- Three-node linear triangle
- Six-node quadratic triangle
- Quadratic (order of solution) Elements
- Eight-node “serendipity” quadrilateral
- Nine-node “Lagrangian” quadrilateral

Water surface and ground elevations are interpolated linearly within each element based on values at the corner nodes. Velocity is interpolated using a quadratic approximation based on values at all the nodes of the element. The quadrilateral elements use identical linear interpolation functions, but their quadratic functions differ because of the presence of an additional node at the center of the nine-node quadrilateral element.

6.7.3 Boundary Conditions and Loading

The global coordinate system of the composite cylindrical shell is defined in such a way that the bottom face of the cylinder lies in the x-y plane and the positive Z-axis is aligned with the axis of the cylinder. The following boundary conditions were imposed on the composite cylindrical shell.

1. The circumferential and radial displacements „v‟ and „w‟ respectively equal to zero at both faces of the cylinder (at z=0 and z=h, v=w=0).
2. Axial displacement „u‟ is zero at the bottom face of the cylinder but is non-zero at the top face where the load is applied (at z=0,u=0 and at z=h, u≠0).

A unit force was applied at the upper rim of the cylinder (z=h) in buckling analysis.

6.7.4 Solution

Linear buckling analysis in ANSYS finite-element software is performed in two steps.

1. In the first step a static solution to the composite cylindrical shell is obtained. In this analysis the prebuckling stress of the shell is calculated.
2. The second step involves solving the eigenvalue problem given in the form of Equation. This Equation takes into consideration the prebuckling stress effect matrix [S] calculated in the first step.

$([K] + \lambda i\ [S])\ \{\psi\}i = \{0\}$

Where [K] = stiffness matrix

[S] = stress stiffness matrix

λi = ith eigenvalue (used to multiply the loads which generated [S])

Ψi = ith eigenvector of displacements.

The "Block Lanczos" method was used to extract the eigenvalues. The eigenvalues obtained from the buckling analysis are the minimum buckling loads.

6.7.5 Analysis

Finite elements analysis was performed on a composite shell without stiffeners having the geometric properties. By obtaining the minimum deformation at maximum load and considering that as a best angle to obtain the minimum buckling load, different analysis were carried out with angle-ply laminate arrangements, composite materials with different properties as shown in section 4.9. The observations made on this analysis are presented in the results and discussions chapter.

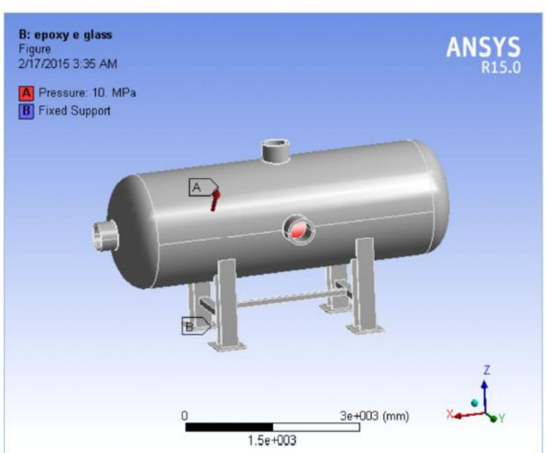


Fig.6.2. loading and BC

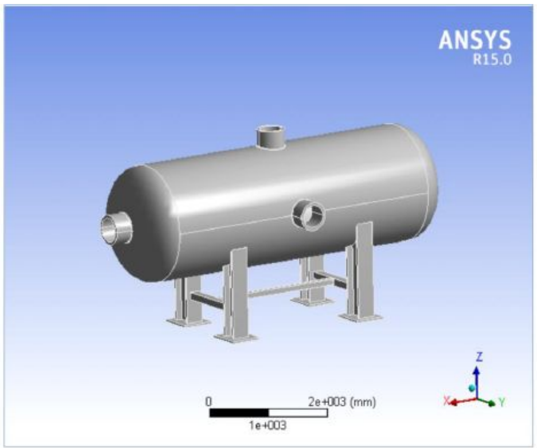


Fig.6.3. Designed epoxy Model

VII. RESULTS

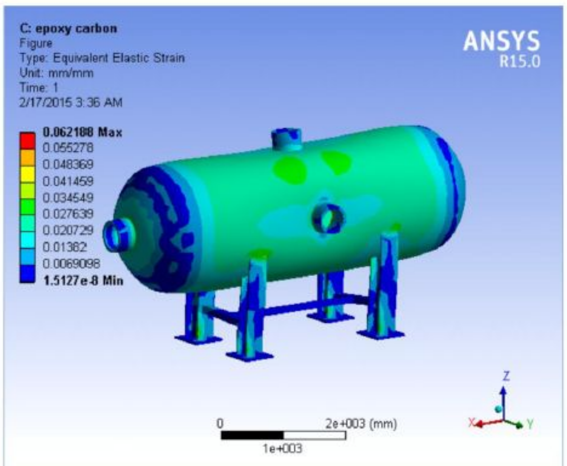


FIGURE 3
Model (C4) > Static Structural (C5) > Solution (C6) > Equivalent Stress > Figure

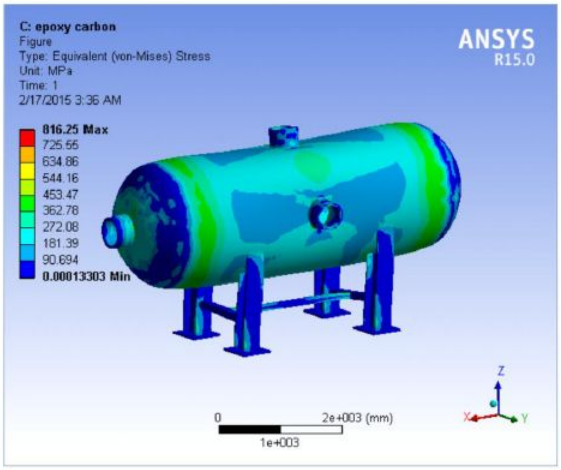
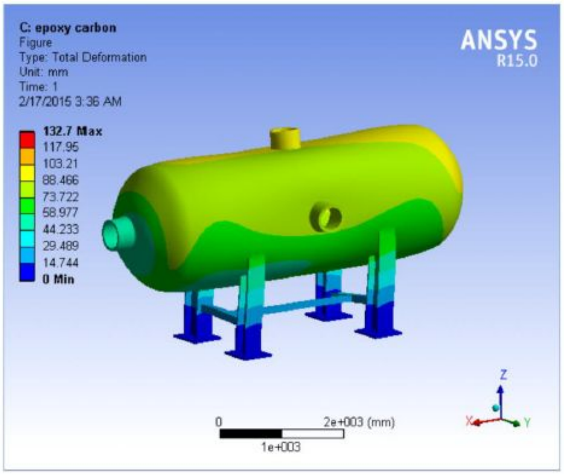


FIGURE 4
Model (C4) > Static Structural (C5) > Solution (C6) > Total Deformation > Figure



GLASS EPOXY Stresses and Deformations

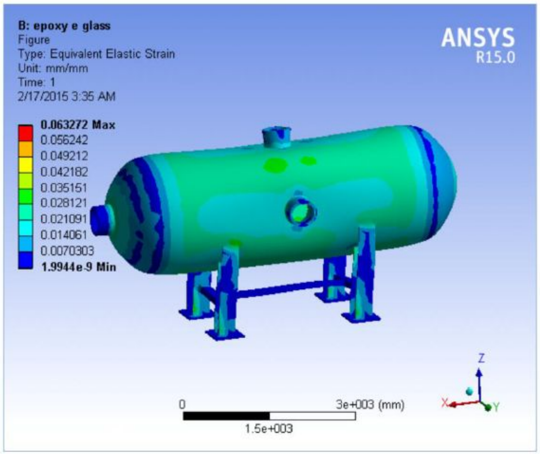


FIGURE 5
Model (B4) > Static Structural (B5) > Solution (B6) > Equivalent Stress > Figure

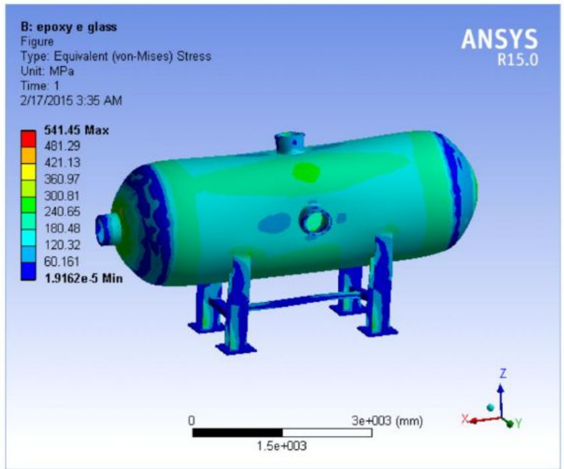
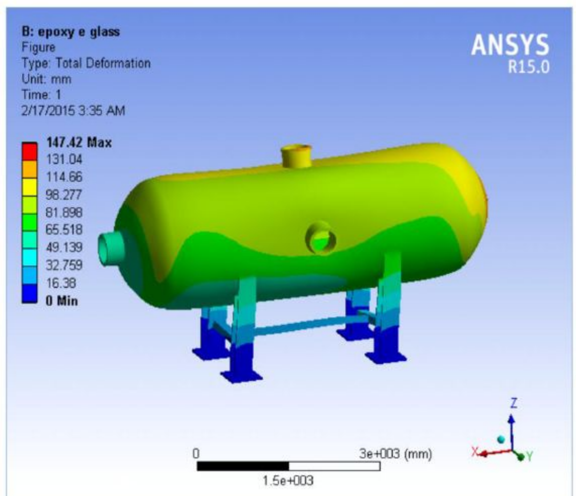


FIGURE 6
Model (B4) > Static Structural (B5) > Solution (B6) > Total Deformation > Figure



VIII. CONCLUSIONS

In this paper, the Finite element analysis of a composite hydrogen storage vessel based on "unit load method" along with complete structural analysis and evaluation of fatigue lifetime were conducted using ANSYS package. The numerical simulations were performed on a fuel cell vehicle's composite high-pressure storage vessel. The following results were obtained:

The vessel is modeled using unit load method under various internal pressures. By increasing the vessel pressure acceptable and appropriate behavior was observed in strain-pressure curves. This indicates high safety of this method in the design of storage vessels.

The aluminum liner plays a fundamental role in design and performance of composite high-pressure hydrogen storage vessels.

Therefore, another method is needed for selection of the appropriate material regarding the geometry and loading of vessel.

The results showed that the fatigue lifetime of vessel depends on the finite element mesh size, crack density and ratio in an element, cyclic loading amplitude and stress status at the liner.

ISSN 2348 – 8034
Impact Factor- 3.155

REFERENCES

1. *Pressure Vessel Design, fabrication and test, ASME section VIII, div. 1*

2. *Ming-Hsien Lu, Jiun-Shya Yu, Jien-Jong Chen, The effect of analysis model on the stress intensity calculation for the nozzle attached to pressure vessel under internal pressure loading, International Journal of Pressure Vessels and Piping 117-118 (2014) 9-16.*

3. *Shafique M.A. Khan G, Stress distributions in a horizontal pressure vessel and the saddle supports, International Journal of Pressure Vessels and Piping 87 (2010) 239-244.*

4. *A.M. Birk G, Scale considerations for fire testing of pressure vessels used for dangerous goods transportation, Journal of Loss Prevention in the Process Industries 25 (2012) 623- 630.*

5. *Drazan Kozak1, Ivan Samardzic2, Antun Stoic, Zeljko Ivandic, Darko Damjanovic, Stress Analyses of Cylindrical Vessel with Changeable Head Geometry, BULETIN ştiinţific, Seria C, Volume XXIII, ISSN 1224-3264.*

6. *Dr. Clemejns Kaminski, Stress Analysis & Pressure Vessels, University of Cambridge, 2005.*

7. *David Heckman, Finite element analysis of pressure vessels, MBARI 1998.*

8. *Michael A. Porter, Dennis H. Martens, Comparison of Linear and Nonlinear FE Analysis of a Typical Vessel Nozzle,199.*

9. *Pavo Balicevici et.al, Strength of Pressure Vessels with Ellipsoidal Heads, Journal of Mechanical Engineering 54(2008)10, 685-692.*

10. *Nidhi Dwivedi and Veerendra Kumar, Burst pressure Prediction of Pressure Vessel using FEA, International Journal of Engineering Research & Technology (IJERT), Vol.1 Issue 7, September-2012, ISSN: 2278-0181.*