International Journal of Interdisciplinary Innovative Research &Development (IJIIRD)
ISSN: 2456-236X
Vol. 05 Issue 02 | 2021

Finite Element Analysis Based Structural Analysis of Horizontal Tube Sheet

Neethu Krishnakumar Nair[1], Dr. S. Ram Reddy[2], Amar kumar[3]

[1]-Student of ME in Machine Design. BRHCET-Vangani
[2]Principal ,SLRTCE ,Mira Road (E)
[3]Assistant professor, Mechanical Engineering department, BRHCET-Vangani

ABSTRACT

Tube sheets serve multiple purposes, either they act as support for filter elements or for connecting tubes for Heat exchangers. Due to its interaction with pressure vessel and the resulting stress that it generates, the design of tube sheet becomes extremely complex. The location where the tube sheet is attached, radial expansion of the vessel is halted; this creates bending stresses in the vicinity of the tube sheet hence the resulting stress profile becomes increasingly complex. Vertical Pressure Vessels are ideally used as they are efficient for tube sheet application. But in case of space constraint, it has to be adjusted and tube sheet is placed horizontally. Due to this an uneven stress distribution is created. The delta pressure on the bottom side is more than on the top side since gravity effect ensures clogging at the bottom more than at the top. Hence, along with axial push the tube filter sheet will also have a slight bending component, and this will create a non-uniform stress profile on the entire assembly. Objectives is to use FEA and determine limits on the non-uniform stress profiles for the operating conditions, so that alarms and safety equipment can be designed according to the outputs.

KEYWORDS:- Heat transfer, Effectiveness, corrugated tube, Plain Tube

.

1. INTRODUCTION

A tube sheet is sheet, a plate, or bulkhead which is perforated with a pattern of holes designed to accept pipes or tubes. These sheets are used to support and isolate to tubes in heat exchangers, filter and boilers support elements. Depending on the application, a tube sheet may be made of various metals or of resin composites or plastic or steel.. A tube sheet may be covered in a cladding material which useful for a corrosion barrier, temperature and pressure effects and insulator and may also be fitted with a galvanic anode. Tube sheets may be used in pairs in heat exchange applications or singularly when supporting elements in a filter.

The studies of existing system in pressure vessel one or two tube are used with small size vessel. Here in this project is totally new design that is proposed there are two tube sheets at equal intervals and combination of three pressure vessel. In this design arrangement of tube-sheets are equally spacing distance and vessel size will be large as compare to existing .design of all model by using ASME Code Section-8, Div-2. Finite Element Analysis based Structural analysis of Horizontal Tube Sheet Filters final result is space, stress, and weight optimization, and as per ASME Code design will be safe for that condition and cost will be a reduces.

This project deals with the Structural analysis of Horizontal Tube Sheet Filters. A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. In here new design of combination of two pressure vessels and two tube-sheets are mounted. Determination of the space between the tube-sheet which is widely used in the filters as main supporting elements of the filter tubes. Pressure vessel are used to store and transmit liquids, vapors, and gases under pressure in general. For analysis purpose static structural are used for model is safe for this condition and optimization of space, stress, weight and also model is safe for this condition as per ASME. the ASME codes are referred. In the primary phase the calculation of dimensions of the tube-sheet were of prime importance. The calculated dimensions of the tube-sheet were then confirmed from the client for further process. In secondary phase the tube-sheet was analyzed and optimized in static structural module for proper convergence and then was analyzed and optimized Tube-sheet module finalized for pick values of stresses and deformation. These stress and deformation values of tube-sheet are used for spaced sequential optimization of the tube-sheet in pressure vessel. The last phase of the project concludes with the validation of FEA results by comparing the FEA results of hydro test in the virtual environment and the experimental results obtained from actual hydro test of the tube-sheet conducted at the client's end.

1.1 Problem Statement

Ideally Vertical Pressure Vessels are efficient for tube sheet applications, however in certain cases of space constraint, it is imperative to adjust and place a tube sheet horizontally. However this creates an uneven stress distribution. As the filter sheets start clogging, gravity effects ensure that the sheets at bottom start clogging more than the ones on the top. This in turn means that the delta pressure on the bottom side is more than on the top side. Hence, along with the axial push that the filter sheet will experience, it will also have slight bending component

1.2 Objectives

The literature survey carried out during the present course of work clearly shows that there is scope for Finite Element Analysis based Structural analysis of Horizontal Tube Sheet Filters Hence the objectives of the present work are decided as under:

- Mechanical Design of Tube-sheet using ASME code.
- To optimize number of supports
- To study the effect of tube sheet spacing on stress profile
- To optimize the structure with the following criteria
    i. Spacing Distance between tube sheet
    ii. Thickness of the tube sheet

1.3 Project scope

- Optimization of Space
- Optimization of stress
- Optimization of weight
- As per ASME Code vessel is safe for that condition

1.4 -Process Flow chart

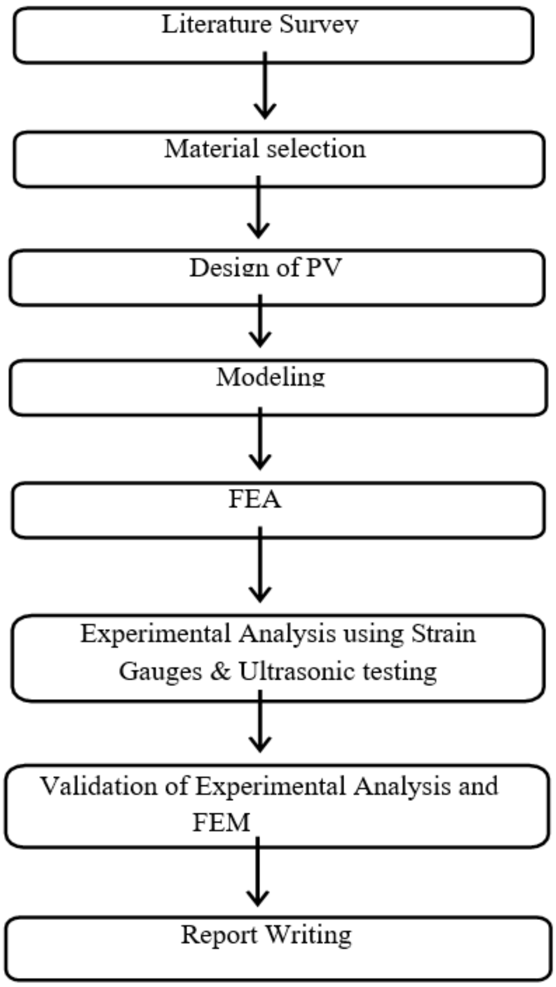


Figure 1:-Process flow chart

2 FEA ANALYSIS

2.1 Model for FEA Analysis

First to create solid IN Workbench model by using all above calculated parameters and ASME Code Section-VIII, Div-II, and after meshing solid model more nodes are generated but capacity of system not sufficient, so create next model in surface. In surface modelling no. of nodes are decreases 40-60 % as compare to solid model, which are under the capacity to my system.

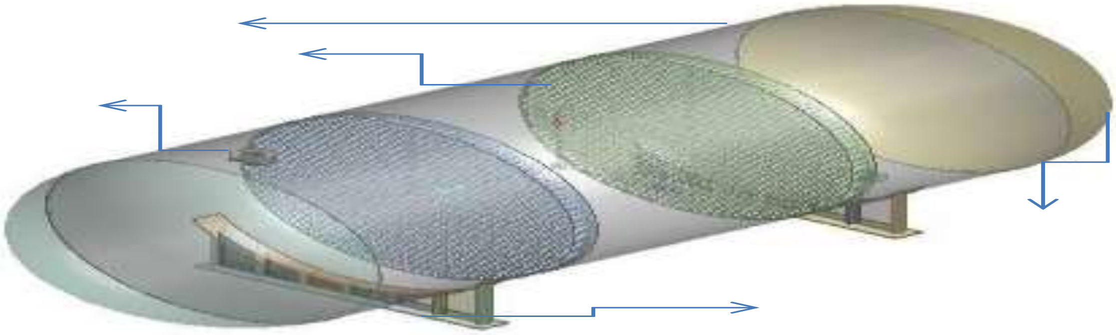


Figure 2. Cad model of horizontal heat exchanger

2.2 Material Properties

Material used for the pressure vessel is SA516GR70 Steel. The properties of the material are described in following table. By using Reference of ASME Code Section- VIII, Div-I.

Table:-Properties of Material

Sr.no.	Description	Quantity
1	Yield Strength	240 MPa
2	Ultimate strength	460 MPa
3	Max allowable stress	138 MPa
4	Modulus of Elasticity	200 GPa
5	Thermal conductivity	60.5 W/mk
6	Poisons ratio	0.3

2.3 Meshing

Basic methodology of finite element method is to prepare calculations at only limited number of points and then interpolate the results for entire domain i.e. surface and volume. Any continuous object has infinite number of degrees of freedom and it is impossible to solve it for required results in this format. Finite element method reduces degrees of freedom from infinite to finite with the help of Discretization of entire domain. Various types of elements like 1D, 2D, 3D, mass, spring, damper, gap etc. are available for meshing, one has to select them depending upon the geometry, size and shape of the component, type of the analysis to be carried out and time availability for completion of project

2.4 Steps Involved in Meshing Pressure Vessel in ANSYS:

- Selection of element type for meshing.
- Creating simplified parts & meshing the parts

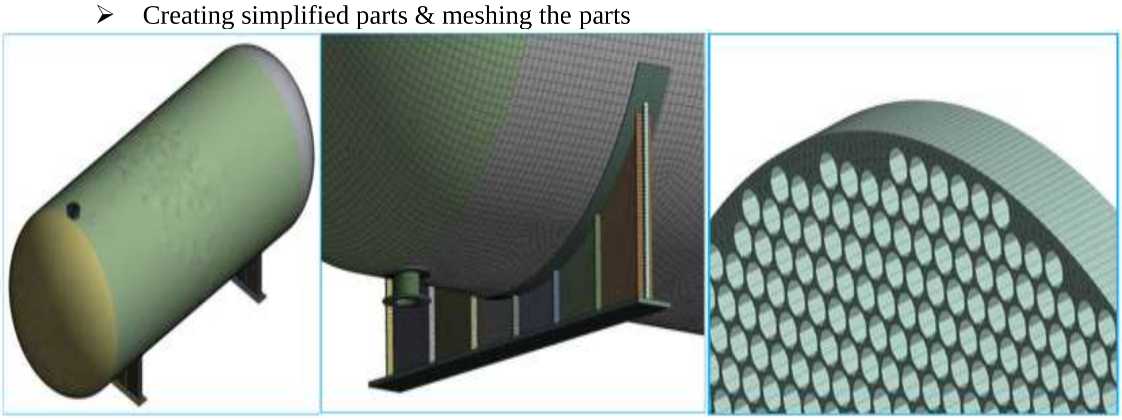


Figure 3:-Mesh Modified Model along with other parts

Table 2:-Element and nodes of various parts

Parts of modal	Method	Element Size	No. of Nodes
Shell	Multiquad/tri	100	34027
Dome(2 nos.)	Multiquad/tri	100	16207
Tube-sheet(2 Nos.)	All triangles	80	16644
Nozzle part(2Nos.)	Quadrilateral dominent	20	1274
Saddle	Quadrilateral dominent	24	11299
			Total = 79451

2.5 Modal Analysis

Applying Boundary Conditions are as follows

- ➢ One saddle support is fixed and check the contact between face to face contacts, edge to edge contacts, edge to face contacts are properly detected or not.

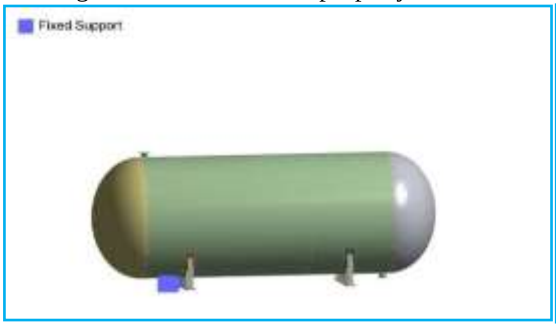


Figure 4 Boundary Condition

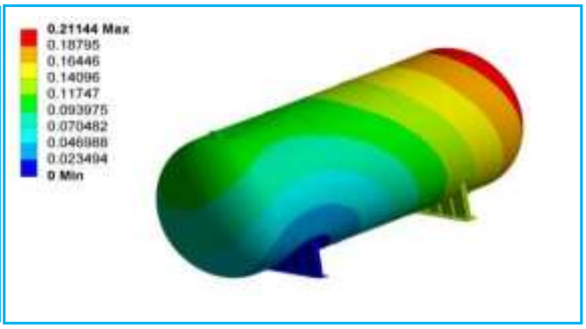


Figure 5 Deformation result

Maximum Deformation = 0.21 mm and Frequency of 1st Mode = 2.6922 Hz, Above results shows maximum deformation is 0.21 mm and Frequency is 2.6922 Hz, so frequency is more than zero shows the all the contacts are properly

2.6 Static Structural Analysis

Applying Boundary Condition Are As Follows, For Pressure Vessel Analysis With Self Weight:

- ➢ Both saddle supports are fixed.
- ➢ Gravity acting downwards.

.

Figure 6:-. Boundary Condition

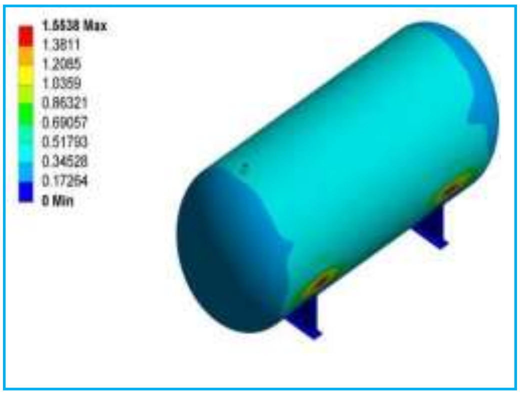


Figure 7:-Maximum Deformation=1.5538 mm

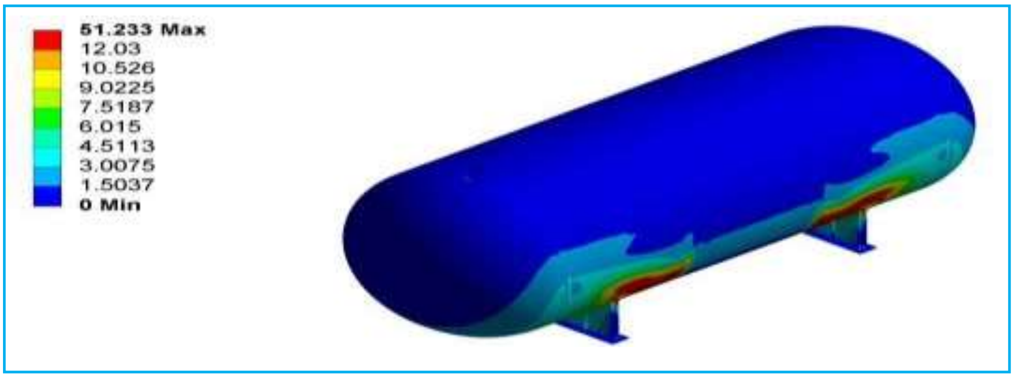


Figure 8:-Result of Deformation

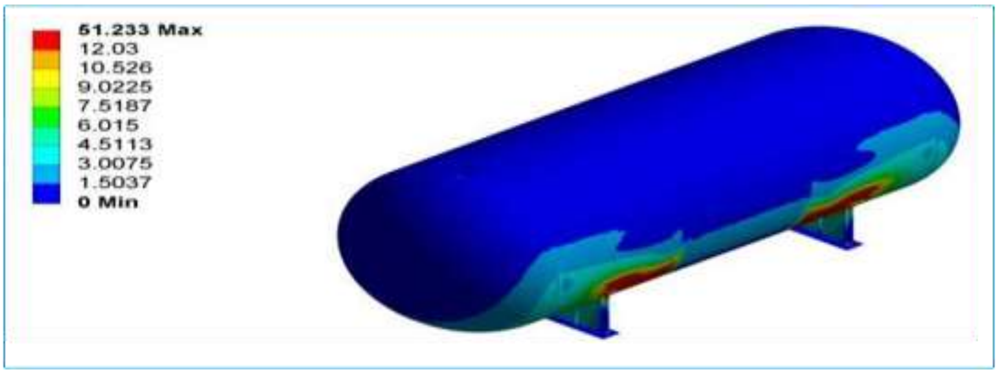


Figure 9 :- Maximum Stress=51.233 MPa.

Above gravity results shows maximum deformation is 1.5538 mm and maximum stress is 51.23 Mpa.at these vessel is safe for self-weight.

Table 3:-Result of stress and Deformation

Sr. NO.	NO. Of Nodes	Stress(Mpa)	Deformation(mm)
1	48537	51.235	1.5536
2	68042	54.945	1.6856
3	88837	59.735	1.8035
4	109362	66.837	1.6501
5	131928	62.176	1.6997
6	151985	64.592	1.6699
7	172904	60.979	1.6841

3. STATIC STRUCTURAL ANALYSIS

3.1 Case-01: Applying Boundary Condition Are As Follows

- Apply Internal Pressure=0.32.
- One saddle support is fixed and on second saddle support apply displacement.

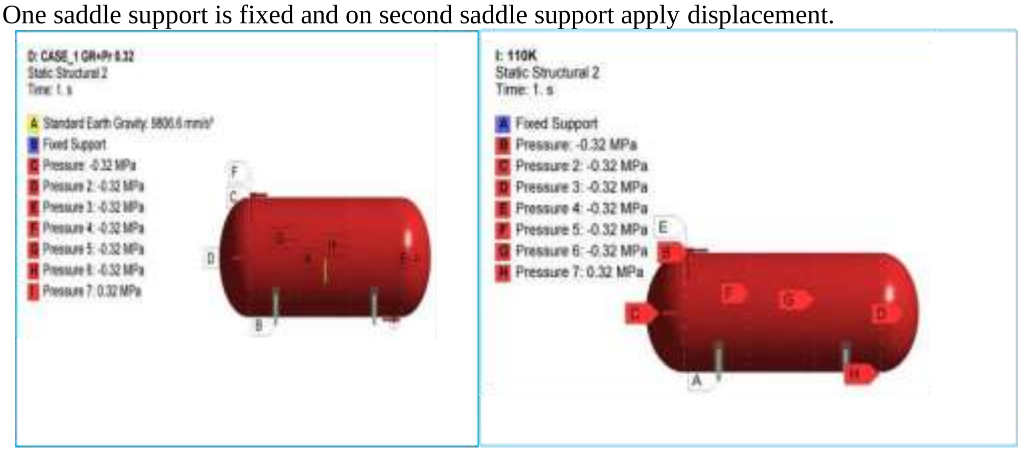


Figure 10:-Boundary conditions for Case 1   Figure11:-Boundary Conditions of fixed support

Applying Internal Pressure of 0.32 MPa on all bodies of Pressure Vessel.

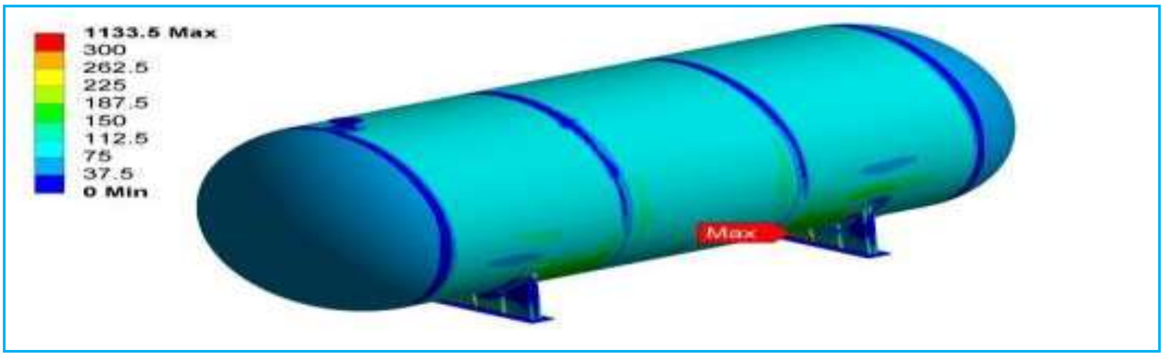
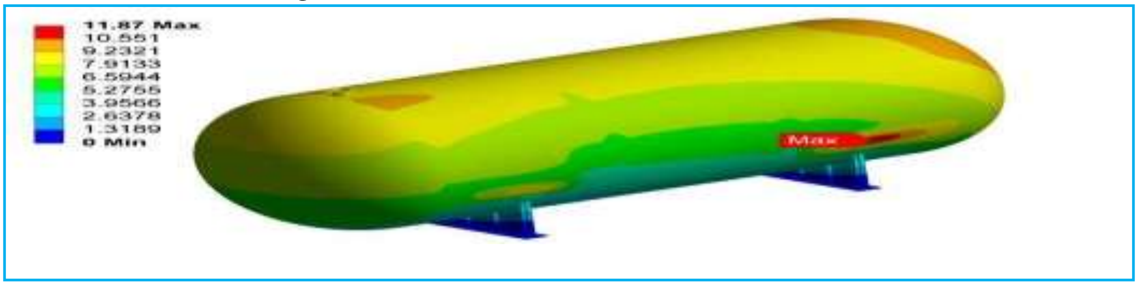


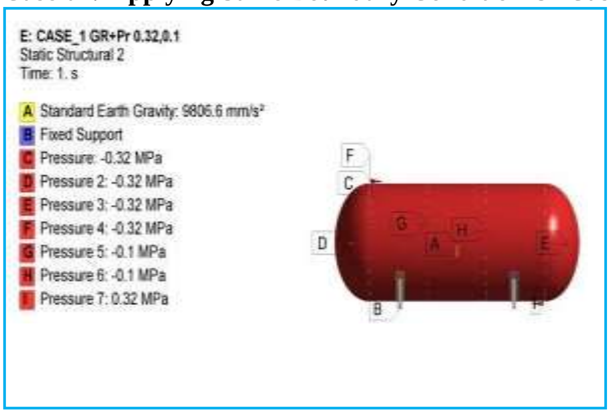
Figure 12:-Result of deformation & stress

Maximum Deformation= 11.87 mm Maximum Stress= 1133.5 MPa

Figure 13:-Result of deformation & stress



Case 02:-Applying Same Boundary Condition Of Case-01 And With Standard Earth Gravity



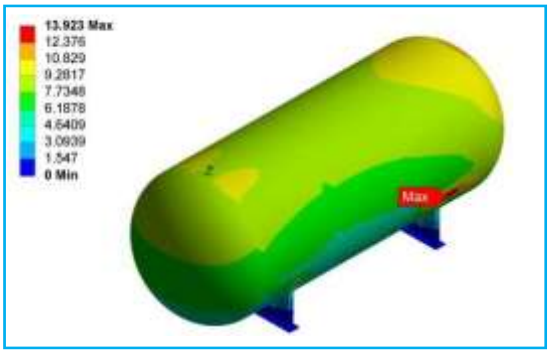


Figure 14:- Boundary Condition.   Figure 15:-Result of Analysis

Case-03: For Same Boundary Condition Only Change Internal Pressure Only On The Tube-Sheets For Measuring Stress Value.

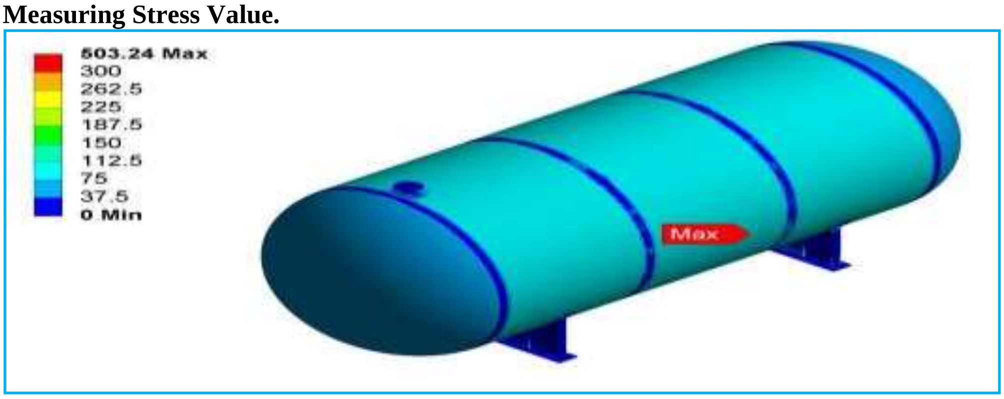


Figure 16:-Result of Case 3

Above test is only for measuring stress with changing tube-sheet pressure, result shows stress is 503.24MPa.

Case 04: Solid Tube-sheet Optimization at Pressure 0.32 MPa

Solid 187 is been used in order to mesh the Ligament model. it is higher order3-D ,10 node element. It has quadratic displacement behavior and it is well suited for modeling irregular meshes. The element is defined by 10 nodes having 3 degree of freedom at each node: translation in nodal x, y, and z direction. The element has plasticity, hyper elasticity, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformation of nearly incompressible elasto plastic material, and fully incompressible hyper-elastic material. The geometry, nodes location and the coordinate system is shown in the below figure. In addition to the node the element data includes the orthotropic or anisotropic material properties. Orthotropic or anisotropic material direction corresponds to the element coordinate directions.

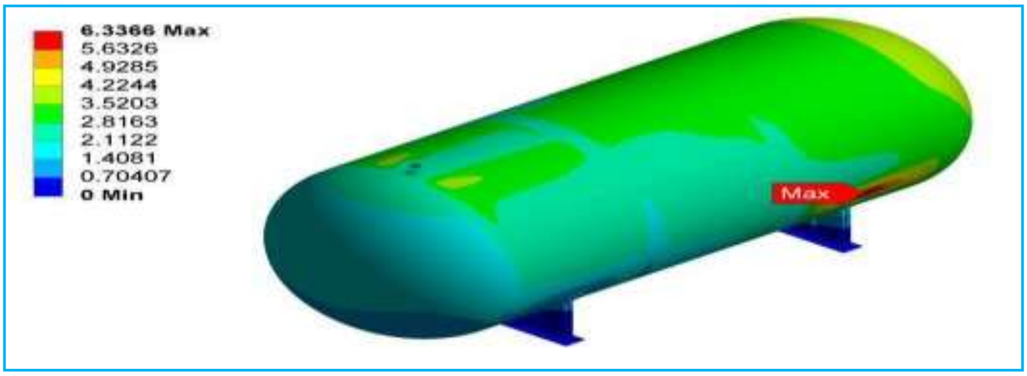


Figure 17:-Result of Case 4

Case 05. Applying boundary condition for optimization of single tube-sheet thickness

- Fixed support
- Apply pressure=0.32MPa

Table 4:-Result of stress and deformation

Sr. NO.	Thickness (mm)	Stress (MPa)	Deformation (mm)
1	409	37.967	0.8955
2	40400400	43.824	1.1079
3	330	61.035	1.6707
4	280	86.032	2.7041
5	240	111.36	4.2581
6	230	121.34	4.8248
7	225	126.83	5.1478
8	220	132.67	5.5009
9	215	138.7	5.8861

Complete Vessel Analysis Considering All Tubes Masses, Pressure On Tube-Sheets 0.01 MPa And Internal Pressure On All Other Components As follows.

Total number of Tubes = 1131

Total number of tube sheets = 2

Mass of each tube = 2.5 kg

Total mass of all tubes on the three tube sheets = 2.5*1131*2 = 5655 kg

Figure 18:-Pressure Vessel with 2 Saddle Supports

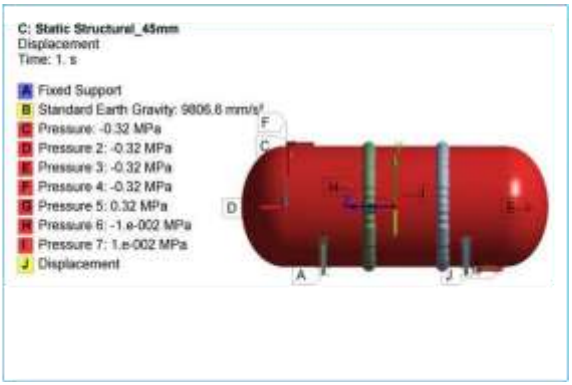


Figure 19:-Supports & Pressure points

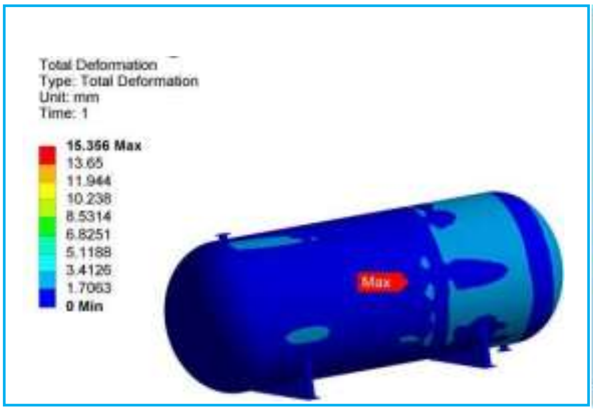


Figure 20:-Total Deformation results

Figure 21:-Result of Equivalent stresss

In the above analysis case the stress is exceeding but is near the allowable stress. So to reduce the stress one more saddle support is suggested.

Case 6:-Complete Vessel Analysis Considering All Tubes Masses, Pressure On Tube-Sheets 0.01 MPaAnd Internal Pressure On All Other Components As0.32 MPa With 3 Saddle Supports.

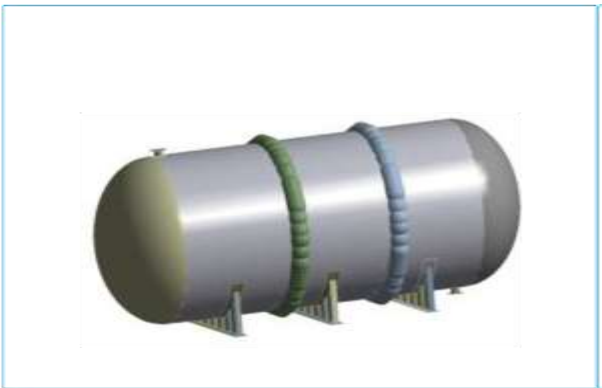


Figure 22:-Pressure Vessel with 3 Saddle Supports

Figure 23:-Mesh Modified Model

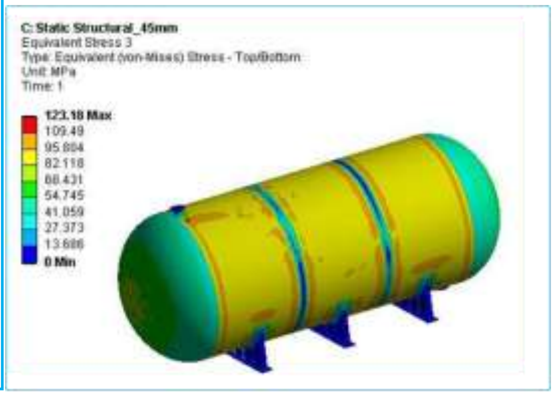


Figure 24:-Maximum Deformation=15.344 mm   Figure 25:-Maximum Stress=123.18 MPa

Stress Result for Complete Vessel Analysis considering all tubes masses, pressure on tube-sheets 0.01MPa and internal Pressure on all other components as 0.32 MPa with 3 saddle supports is 123.18 MPa which is within the limit of allowable stress. Now the complete Pressure Vessel considering all the boundary conditions and pressure conditions is Safe, as the stress is within the allowable limit of 138 MPa.

4. RESULTS AND DISCUSSION

- In Modal analysis contacts are properly detected.
- Static Structural for Gravity test Model Safe for self-weight.
- From nodes Vs Stress graph select pick point (109362) no. of nodes where solution is converging as per mesh sensitivity.
- Tube-Sheet optimization for weight reducing, 230 mm thickness of tube-sheet is selected. And reduced weight of Tube-Sheet from 13988 kg to 2927.7 kg.
- Tube-sheet optimization including point mass weight of 2.5 kg of each tube for reducing weight and material, 45 mm of tube-sheet is finalised and reduced weight up to 20 %
- Design of 2 saddle supports was not stable hence based on FEA results, suggestion of 3 supports was recommended.
- Stress Result for Complete Vessel Analysis considering all tubes masses, pressure on tube-sheets 0.01MPa and internal Pressure on all other componentsas 0.32 MPa with 3 saddle supports is 123.18MPa which is within the limit of allowable stress. And maximum deformation of 15.344 mm on tube sheet and linear movement of vessel 1.7048 mm.

5. EXPERIMENTATION ANALYSIS AND VALIDATION

HBM Sensor provides complete measurement solutions – from sensor to software – for industrial and laboratory applications. These include: Strain gages Transducers for weight, force, pressure, torque and displacement Amplifiers and signal conditioning electronics Data acquisition systems, including software. In this test show no leakage is detected By using strain gauge (LC4C1X) HBM Sensor and deformation result are as follows**.**

FEA RESULT	EXP. RESULT	% ERROR
15.344 mm	14.09 mm	8.17

Figure 26. Result comparison

5.1 Ultrasonic Test

Ultrasonic Testing (UT) uses high frequency sound energy to conduct examinations and make measurements. Ultrasonic inspection can be used for flaw detection/evaluation, dimensional measurements, material characterization, and more. To illustrate the general inspection principle, a typical pulse/echo inspection configuration as illustrated below will be used.

A typical UT inspection system consists of several functional units, such as the receiver, transducer, and display devices. A pulser/receiver is an electronic device that can produce high voltage electrical pulses. Driven by the pulser, the transducer generates high frequency ultrasonic energy. The sound energy is introduced and propagates through the materials in the form of waves. When there is a discontinuity (such as a crack) in the wave path, part of the energy will be reflected back from the flaw surface. The reflected wave signal is transformed into an electrical signal by the transducer and is displayed on a screen. In the applet below, the reflected signal strength is displayed versus the time from signal generation to when a echo was received. Signal travel time can be directly related to the distance that the signal traveled. From the signal, information about the reflector location, size, orientation and other features can sometimes be gained.

Figure 27:- Pulser for Ultrasonic testing

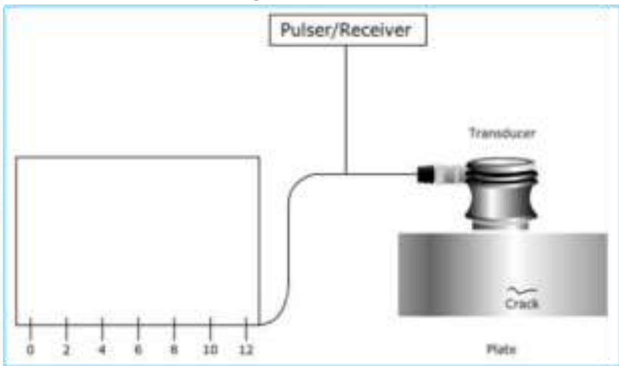


Figure 28:-Systematic diagram of UT Testing

Ultrasonic Inspection is a very useful and versatile NDT method. Some of the advantages of ultrasonic inspection that are often cited include

- It is sensitive to both surface and sub surface discontinuities.
- The depth of penetration for flaw detection or measurement is superior to other NDT Methods.
- Only single-sided access is needed when the pulse-echo technique is used.
- It is highly accurate in determining reflector position and estimating size and shape.
- Minimal part preparation is required.
- Electronic equipment provides instantaneous results.
- Detailed images can be produced with automated systems.

It has other uses, such as thickness measurement, in addition to flaw detection

5.2 Steps for Working

Step 1: The UT probe is placed on the root of the blades to be inspected with the help of a Special bore scope tool (video probe).
Step 2: Instrument settings are input.
Step 3: The probe is scanned over the blade root. In this case, an indication (peak in the data) through the red line (or gate) indicates a good blade; an indication to the left of that range indicates acrack.
A probe sends a sound wave into a test material. There are two indications, onefrom the initial pulse of the probe, and the second due to the back wall echo. RIGHT: Adefect creates a third indication and simultaneously reduces the amplitude of the back wall indication. The depth of the defect is determined by the ratio *D*/*E*p.

6. CONCLUSION

- The project is basically focused on an Analysis and optimization of space sequential tube-sheet in pressure vessel. Design of pressure vessel are done by ASME Code Section-8,Div-2.
- The Analysis of pressure vessel model was done in ANSYS 15.0 workbench. The results were supported with an experimental validation for verifying the actual deformation and FEA results. Following are concluding remarks based on the analysis performed on vessel.
- Firstly analysis of pressure vessel model is done to develop the standard operating procedure. From the comparison of results at different mesh size.it is concluded that variation in results is within acceptable limit, hence approximately 100000 nodes mesh size is fixed for further analysis.in that maximum stress is 66.837MPa and deformation is 1.6501 mm.
- Result of optimization of Tube-Sheets we found that Thickness is reduced from 215 mm to 45 mm and also Reduced Weight from 13988 kg to 2927.7kg.
- 5-Tube-sheet optimization including point mass weight of 2.5 kg of each tube for reducing weight and material, 45 mm of tube-sheet is finalised and reduced weight up to 21%.

- 6-Stress Result for Complete Vessel Analysis considering all tubes masses, pressure on tube-sheets 0.01MPa and internal Pressure on all other components as 0.32 MPa with 3 saddle supports is 123.18 MPa which is within the limit of allowable stress.
- 7-Above all the conclusion shows the optimization of stress, space, and optimization of tube-sheets is reduced weight up-to 21 % and also reduced material is done. For this condition model is safe as per ASME Section-VIII, Div-II. and FEA results and Experimental results are in close resemblance and proved that FEA analysis is correct and is validated by experimental deformation results Manufactured tested values are within 20% of FEA for large vessel calculations ,hence our FEA results 8.17% are reliable. No leakage and No damage is detected by using Strain Gauge Sensor and ultrasonic testing machine. Finalized vessel satisfies ASME Criteria and this has been validated through FEA.

7. FUTURE SCOPE

- Changing the material used for the Tube sheet – Comparatively stronger material can be used and analyzed for the fatigue life. The thickness of tube sheet required in this case will be comparatively less.
- Reducing the number of filter holes in the tube sheet. – Reducing the number of holes may increase the ligament efficiency of the tube sheet. Further analysis can be done using Thermal Analysis

8. REFERENCES

[1] H.F. Li,Qian, C. F., & Yuan, Q. B. (2010). Cracking simulation of a tube sheet under different loadings. Theoretical and Applied Fracture Mechanics, 54(1), 27- 36.
[2] V. G. Ukadgaonker, P. A. Kale, Mrs. N. A. Agnihotri& R; ShanmugaBabu (1995). Review of analysis of tube sheets. Int J. Pres. Ves. & Piping 61(1996) 219-297.
[3] SwetaNaik Thickness and shape optimization of Filter Tube-sheet by Non Linear FEA' [Journal]. - [s.l.] : Global Journals Inc. (USA), 2013. - 1 : Vol.13.
[4] W. J. O'Donnell B. F. Langer 'Design of Perforated Plates' [Journal]. - [s.l.] : ASME, 1962. - Vol.84.
[5] ShugenXu, Yanling Zhao. Using FEM to determine the thermo-mechanical stress in tube to tube–sheet joint for the SCC failure analysis. Engineering Failure Analysis 34 (2013)24–34.
[6] Liu Minshan 'Stress analysis of Ω-tube-sheet in waste heat boiler' [Journal]. - [s.l.] : Journal of Pressure Equipment and Systems, 2006.2010
[7] ASME Boiler and Pressure Vessel Codes Section- VIII Division-II.
[8] R. D. Patil, Dr. Bimlesh Kumar. Stress Analysis of Perforated Tube Sheet Used for Pressure Vessel having Square Pitch Hole Pattern. IJIRSET, Vol. 4, Issue 1, January 2015, ISSN (Online): 2319 –8753