International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 1, Vol. 2, March 2015.
e-ISSN: 2394-5141, p-ISSN: 2394-5133, pp 17-23.

Modeling, Stress and Welding Strength Analysis of Pressure Vessel

Ramesh B. T.[1]
Assistant Professor
rameshbt049@gmail.com

Ashok R. Banagar[2]
Assistant Professor
jnnashok@gmail.com

Dr. R. P. Swamy[3]
Associate Professor

[1]Dept. Of Mech. Engg., STJIT, Ranebennur. India

[2]Assistant Professor, Dept. Of Mech. Engg., SMVITM, Bantakal, Udupi. India

[3]Dept. Of Mech. Engg., UBDT College of Engg., Davanagare. India

Abstract **- Stress analysis plays important role in structural optimization and safety of the equipment. Prior estimation of stress helps in preventing failure of the components. In the present work, the pressure vessel has been analyzed for cases of loading. In the first case a dent formation near the nozzle region is considered to find the strength and in the second case full problem is considered for analysis. Initially the dent model is built using shell approach and thickness is assigned as real properties. In the final three dimensional models, nozzles are built as per the drawing and analysis is carried out for all major nozzles as per the load specifications for the problem. The nozzle regions are fine meshed to obtain accurate results as the solution accuracy depends on the finer size of the mesh. The results for von-mises stress are captured due to the ductile nature of the pressure vessel system. The stresses are concentrated near the nozzle opening regions as per the observation.**

Index Terms**—**Pressure vessel, Hoop Stress, FEM, Von Mises Stress.

I. Introduction

A storage tank or container that is designed for operations such as carrying, storing, or receiving fluids are in general, called pressure vessels. A pressure vessel defined as a container The inside pressure is usually higher than the outside Pressure vessels often have a combination of high pressures together with high temperatures, and in some cases flammable fluids or highly radioactive materials. Because of such hazards it is imperative that the design be such that no leakage can occur. In the present work, the pressure vessel has been analyzed for cases of loading. In the first case a dent formation near the nozzle region is considered to find the strength and in the second case full problem is considered for analysis. Initially the dent model is built using shell approach and thickness is assigned as real properties. In the final three dimensional models, nozzles are built as per the drawing and analysis is carried out for all major nozzles as per the load specifications for the problem. The nozzle regions are fine meshed to obtain accurate results as the solution accuracy depends on the finer size of the mesh. The results for von-mises stress are captured due to the ductile nature of the pressure vessel system. The stresses are concentrated near the nozzle opening regions as per the observation.

A. Introduction to Pressure Vessels

Pressure vessels are used in a number of industries; for examples; the power generation industry for fossil and nuclear power[1], the petrochemical industry for storing and processing crude petroleum oil in tank farms as well as storing gasoline in service stations, and the chemical industry (reactors) to name but a few. In nuclear power plant, the figure 1 shows the reactor vessel is a pressure vessel containing the coolant and reactor core [2]. Their use has expanded throughout the world. Pressure vessels are in fact, essential to the petrochemical and nuclear industries

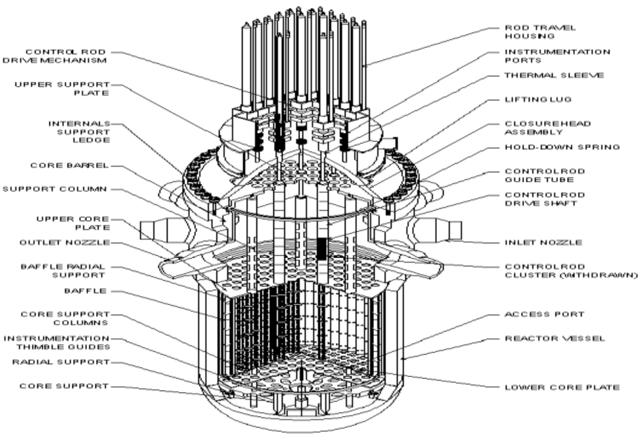

Figure 1 - Pressure Vessel

[8]. It is in this class of equipment that the reactions, separations, and storage of raw materials occur. Generally

B. Applications of Pressure Vessels

Pressure vessels are air-tight containers used mostly in process industry, refinery, and petrochemical plant to carry or hold liquid, gases or process fluids. The commonly used types of pressure vessels in industry are heat exchangers, tanks, towers, boilers, reactors, drums, condensers, piping, etc.

Boilers are pressure vessels designed to heat water or produce steam, which can then be used to provide space heating and/or service water heating to a building. Heat exchangers are devices used to transfer heat energy from one fluid to another.

C. Pressure Vessel Related Accidents

Below are some examples of major accidents involving pressure vessels that have occurred in the past few years,

- A pressure vessel weighing 22680 kg (50,000 pounds) exploded at Marcus Oil
  in 2004, a Chemical plant in Houston, Texas, throwing heavy fragments into the community, which damaged a church, shattered car windows, nearby buildings experienced significant structural and interior damage due to improper modification and faulty welds of the vessel.
- The Buncefield (UK's fifth largest depot) accident on Sunday 12 December 2005 that injured 43 people readily comes to the mind. Twenty petrol tanks were involved in the Buncefield blaze rage; each held three million gallons of fuel. Over 2,000 people were evacuated from the neighbourhood of the depot during the accident..
- Pressure vessel failure in Houston, United States, in the summer of 2008 killed a veteran supervisor when a heat exchanger exploded in a resin-production facility.
- Two employees killed at an oil refinery in southeast New Mexico, USA and two others critically injured after a storage tank exploded into flames on 03 March 2010.

D. Case Studies

Case 1: Analysis of the Catastrophic Rupture of a Pressure Vessel – July 1984 in Chicago

On July 23, 1984, an explosion followed by a fire occurred at a petroleum refinery in Chicago, killing 17 people and causing extensive property damage. NBS was requested by the Occupational Safety and Health Administration (OSHA) to conduct an investigation into the failure of the pressure vessel that eyewitnesses identified as the initial source of the explosion and fire.

The investigation was complicated by the damage caused by the explosion and fire. The explosive force had been sufficient top ropel the upper 14 m of the vessel a distance of 1 km from its original location, while the base remained at the center of the subsequent fire. Sections of the vessel were shipped to NBS in August 1985, where a multi-disciplinary team sought the cause of the failure. Initial And Final Configurations Pressure Vessels show n above Figure 2. More aggressive measurements were then pressurized equipment is required for a wide range of industrial plant for storage and manufacturing process. undertaken to examine the mechanical and chemical characteristics of the initial and replacement components. The cause of failure did not become clear until metallographic results were combined with stress corrosion cracking and hydrogen embitterment tests, followed by a

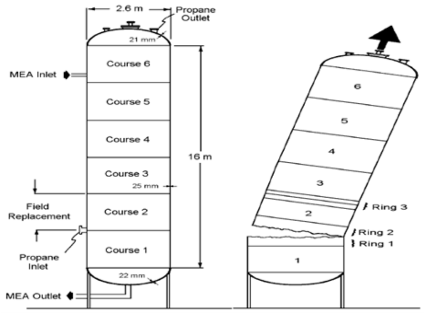

Figure 2 Initial and Final of Pressure vessel

fracture mechanics analysis.

Case 2: Pressure Vessel Failure during hydrotest - November 2007 in China

This vessel was manufactured by a vessel vendor in China and the plate was of Chinese mill origin. Unfortunately this is another example of serious equipment/material failures with equipment being sourced out of the rapidly developing economies such as China, Eastern Bloc and others. These examples like Figure 3 are becoming almost a weekly occurrence now and are exhibiting failure modes not seen in the mature manufacturing economies since the 1930's. This pressure vessel had reached fifty percent of the required test pressure when the shell ruptured possibly due to weld failure.

II. OBJECTIVE OF THE PROJECT WORK

The objectives of the project work are as followed.

• Finite Element Model development with suitable approach

• To determine hoop stress. For this, theoretical calculations are to be carried out.

• To determine hoop stress. For this, ANSY 14.5 software is used.

• To study the comparison of these results.

• To find the structural safety using static analysis of pressure vessel

III. FINITE ELEMENT METHODS

Analysis by using a finite number of elements is the latest method to analyze complicated shapes [8]. Using simple analytical methods analysis of simple structures is done. But to analyze more and more complicated structures, we need to approximate the analytical methods and do various combinations of simple shapes to get the results of the machine member. This method is not accurate, and the errors in the analysis increase at every step. The entire continuum is divided into discrete elements. Analysis is performed on each element. This method is more accurate and gives good result. The process of solving design problems by using finite element methods is a tedious and confusing process, but the results are very much accurate.

A. Analysis Process

The analysis process consists of designing, modeling and analysis. The various stages in the finite element analysis method are: -

- Geometric modelling*:* - is the process of building the model in the database, which defines the contour of the model, and gives it a solid definition.
- FEM modelling*:* - is the stage where the model is meshed by using the desired kind of elements. This forms the primitive for the analysis.
- Constraining*:* - the model by restricting the degrees of freedom of the model to move in the desired direction. This defines the mounting of the model in actual use.
- Loading*:* - of the model to define the forces that act on the model. These forces are the ones that act on the model in actual use, and generate the stresses and strains.
- Post-processing*:* - gives the final result after analysis of the design for the given load boundary conditions by showing the stresses and strains that are generated in the model. The various stresses and strains can be plotted graphically.

Figure 3 - Failure Pictures of Pressure vessels

IV. PROBLEM DEFINITION

The weld Strength analysis of pressure vessel system along with thickness reduction effect on the safety of the pressure vessel. Fatigue analysis is another requirement of the problem for the structural safety.

So the problem objectives include

- Modelling of pressure vessel system
- Meshing and analysis of the pressure vessels
- Finite element modelling of thickness reduction above the nozzle
- Analysis and finding stress at the reduced thickness area
- Complete problem analysis
- Estimation of alternating stress(Fatigue Life)
- Finding stress in the weld regions

A. Material Details:

SA304 is the most versatile and the most widely used of all stainless steels. It chemical composition, mechanical properties, welds ability and corrosion/oxidation resistance provides the best all-round performance stainless steel at relatively low cost. If intergranular corrosion in the heat affected zone may occur [3], it is suggested that SA 304L be used [5].

- Material: stainless steel
- Yield Stress: 335Mpa
- Allowable stress: 170Mpa
- Poison's ratio ν: 0.29
- Young's modulus E: 200Gpa
- Density, ρ: 7900 kg/m3
- Coefficient, of thermal expansion, α: 16.8*10-6 $/^0C$
- Heat conduction, λ: 15 W/m^0C

B. Model Information

The pressure vessel system is created by many nozzle systems varying from smaller diameter to bigger diameters. The Figure 4 shows full model dimensions for the analysis. Only important nozzles are considered for analysis. Diameter of nozzle less then 25mm is not considered for the analysis. Lift pads also represented in the problem. All the dimensions are represented in mm.

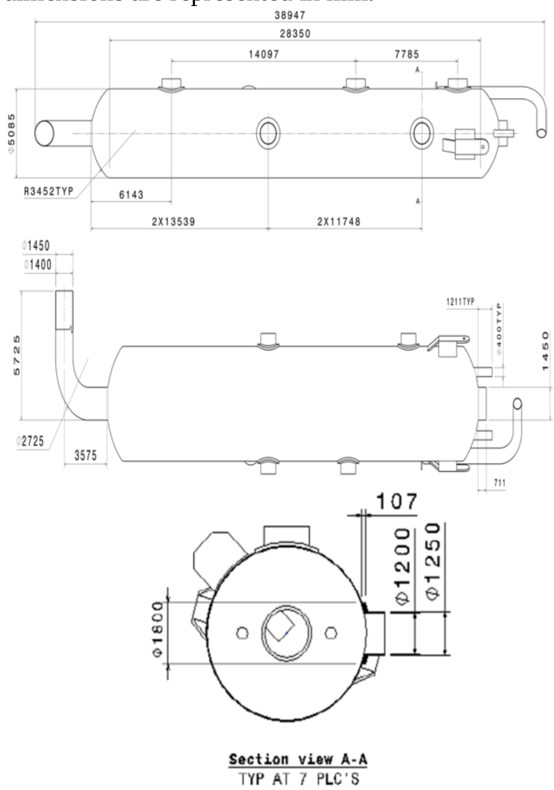

Figure 4 - 2D Geometrical Dimensions of the problem

C. Theoretical Calculations as per ASME standards

Calculations of Shell Thickness as per UG27

- Design Pressure p: 2.1 bar=0.21Mpa
- Allowable stress σ: 170Mpa
- Insider diameter D: 5085mm
- Inside radius R : 2542.5mm

- Joint efficiency E : 0.7 (70% weld strength is assumed)
- When pressure does-not exceed 0.385 σ
  E = 0.385*170*0.7=45.8Mpa
  Since the design pressure of 0.21 MPa is less then 45.8Mpa,
- Minimum required thickness of pressure vessel :

$$\frac{pR}{(\sigma E - 0.6p)} = \frac{0.21 * 2524.5}{(170 * 0.7 - 0.6 * 0.21)}$$

=4.49mm

- But provided thickness is 18mm for the pressure vessel considering hydro-test and other structural loads with buckling considerations. As per ASME standards, the pressure vessel having height more than 10M should minimum thickness of 10mm.
  Hydro-test Pressure Calculations:
- As per ASME standards, hydro test is the maximum pressure in the system.
  Generally hydro test pressure is considered as 1.5times the design pressure.
- So Hydro test pressure p: 1.5*2.1=3.15bar or 0.315MPa Hydro test pressure is always more than the operational pressure. Since
- Thickness of 18mm is considered. So it accommodates the higher hydraulic pressure.
  Minimum Flange thickness calculations:-UG45
  As per the UG45, the neck thickness to the nozzle should be equal to the shell thickness.
- Additional corrosion allowance of 0.06mm should be given.
- Neck thickness required =4.49+.06=4.55mm.
- Thickness considered at the neck region =25mm

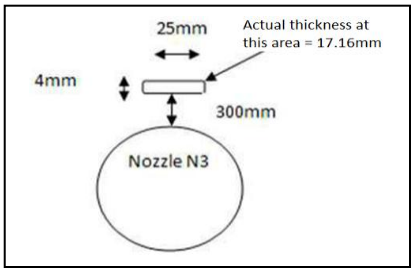

Figure 5 – Location of thin region

The Figure 5 shows reduced cross sectional area and its spread. The dimension is reduced from 18mm to 17.16mm for a dimension of 4mX25mm at a distance of 300 mm from the outer diameter of the nozzle.

D. Geometrical Modeling for Case 1:

Two dimensional models built up using ansys mixed up approach for analyzing the effect of reduction in thickness from 18mm to 17.16mm. A flange is created at the joint of nozzle to the pressure vessel shell and shown in Figure 6. Applied pressure of 2.1 bar on the structure. The entire inner surface is selected and the pressure load is applied at the inner surface. Table 1 shows the type of elements selected.

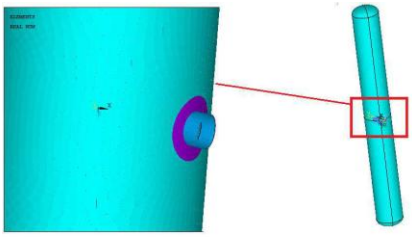

Figure 6 – Geometry built up to analyze the thickness reduction area

Table 1 - Number of Elements & nodes

Element Type	No. of Elements	No. of Nodes
Shell-63	59347	59538

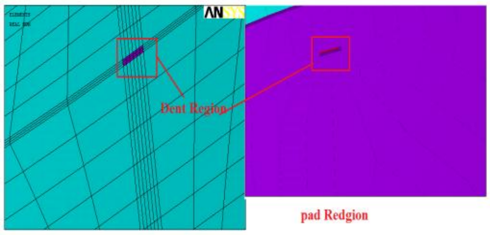

Figure 7 – Under thickness location

E. Thermal Loading Conditions

Thermal difference of 50^0 degrees is applied as the max operating temperature is 73^0 and room temperature is assumed as 23^0.

Case 2: Full Meshing and boundary Conditions details

Brick elements are used for meshing the geometry. Using Hypermesh, the geometry is split and meshed with brick elements for better quality. Aspect ratio, warpage, skew angle, and jacobian are checked for better quality mesh. Nozzles are grouped into separate collectors. Weld regions are separated for checking the stress condition. The figure 8 shows three dimensional meshing of the problem.

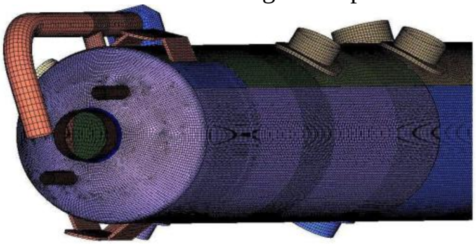

Figure 8 - Mesh view - 3D Brick element

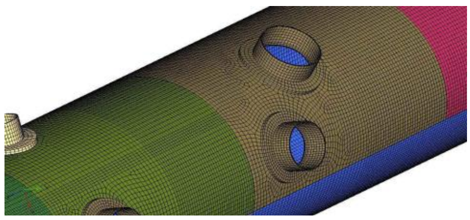

Figure 9 - Magnified mesh views

F. Boundary Conditions

An RBE3 element is created to apply the loads at proper locations. RBE3 element is the most useful element to apply rotational loads to three dimensional elements. Table 2 shows Nozzles loads at different nodes.

Table 2 - Nozzles loads at different nodes.

Nozzle Number	Size (mm)	P (N)	M_C (N-m)	M_L (N-m)	M^T (N-m)	V_C (N)	V_L (N)
N1	610	42000	21000	220500	63000	42000	42000
N2	914	247713	96128	240319	406695	247713	247713
N3	762	55000	23000	62000	85000	55000	55000
N4	914	72600	33000	61600	85000	72600	72600
N5	762	66000	27500	198000	93500	66000	66000
N6	914	243210	94380	235950	399300	243210	243210
N7	203	14000	20000	22000	22000	14000	14000
N8	254	12500	15000	30000	30000	22000	22000
N9	635	45000	62000	68000	34000	45000	45000
N10	762	55000	23000	62000	85000	55000	55000
N11	406	14574	7267	8473	16946	14574	14574

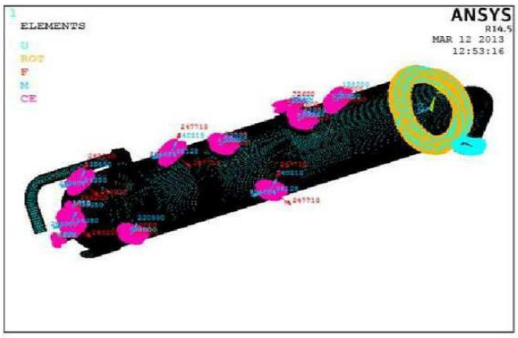

Figure 10 – Applied Boundary Conditions

V. RESULTS & DISCUSSION

The results are represented with two load cases. In the first case, results are represented for maximum loading conditions to find the localization effect of reduction in the thickness. In the next case full problem is analyzed for major nozzle sections and loads along with thermal effects and variation of pressure load effect to find the structural safety. Von-mises is the most popular theory for finding the structural failure of the members. It is based theory like a distortion energy theory and Tresca's theory which represents all combination of stresses like stresses in orthogonal directions and stresses in rotational directions. The analysis is done for two cases.

Case 1: Self weight analysis

Many times self weight plays important role in structural stress generation. Many Structures fail by self weight itself. This can be avoided by proper rib design for the Structure. So initial estimation of self weight effect is also important in the problem.

Case 2: Full loading conditions

This condition is required as the full loading structure has the maximum stresses and Strains. It helps in factor of safety in the problem. It gives guarantee for working Condition of the problem.

A. Effect of thickness reduction on Structural Stability

Figure 11 shows that maximum deformation of 23.3891mm at the top end of the pressure vessel system due to the applied loads. The status bar shows variation of deflection with the color representation. The blue color represents minimum deflection and red color shows maximum deformation. Other colors show variation of displacement.

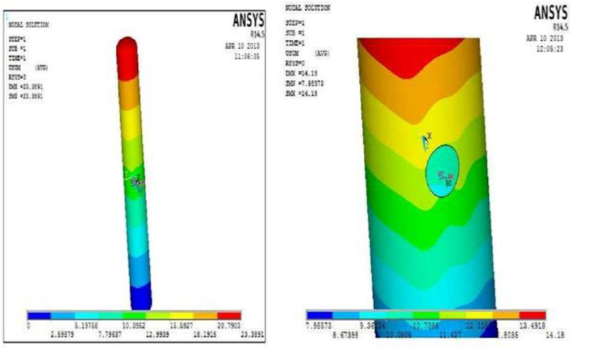

Figure 11 - Overall Displacement Plot for Crude Stabilizer Column and Deformation in the nozzle region.

B. Analysis of Von-Mises Stress in pressure Vessel

The Maximum stress observed is 116.25 MPa. This stress is less than the allowable stress of the structure. So structure is safe for the given loading conditions. Figure 12 shows Von-mises stress is the stress corresponding to stored energy of the system once this stress is more than allowable or critical stresses, the structure is said to be under failure as per the structural analysis. The stress in shell region is 57.7Mpa. This stress is less than the allowable stress of the material. So structure is safe for the given loading.

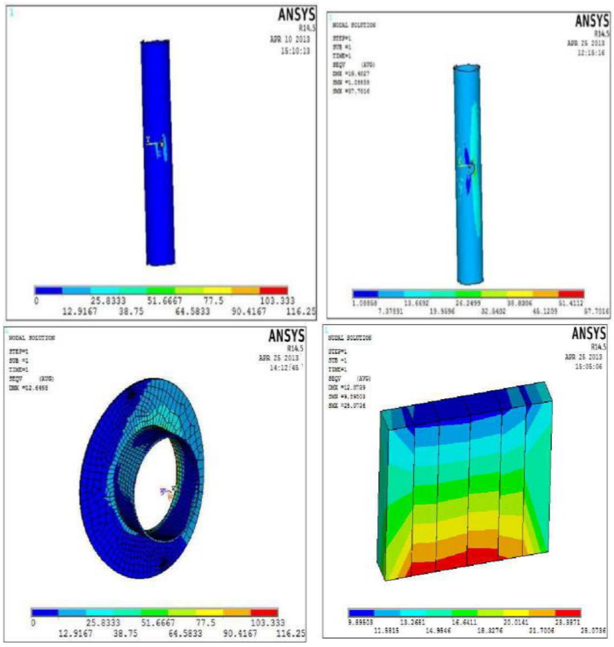

Figure 12 - Von-Mises Plot of Pressure Vessel, in the shell, Nozzle, and thickness location.

C. Full problem Analysis:

The Figure13 shows maximum stress of 136Mpa with all the loading conditions.

Maximum stresses are taking place near the support region. This is clear from the fact that maximum stresses takes place at the fixed location as the case similar to the cantilever beam subjected to end load. The individual component stresses are as follows.

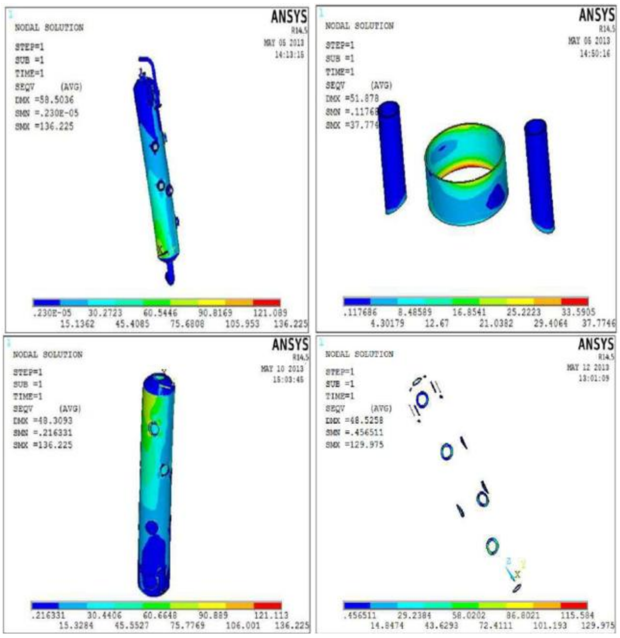

Figure 13 - Von-mises Stress in the top Weld region, nozzle, shell.

The figure 14 shows maximum stress development at the inner boundary to reduced stress at the outer boundary. At some location of the weld stresses are reducing to minimum values.

D. Fatigue analysis

In fatigue analysis fluctuation of pressure load is considered with other loads maintaining constant. The pressure variation is given as 1.5Mpa to 2.1 Mpa. Two load cases are created to apply this load. The stress value is reduced due to reduction in the pressure value. This can be attributed to reduced hoop stress in the structure. Maximum stress of 94.4Mpa can be observed for 1.5bar pressure.

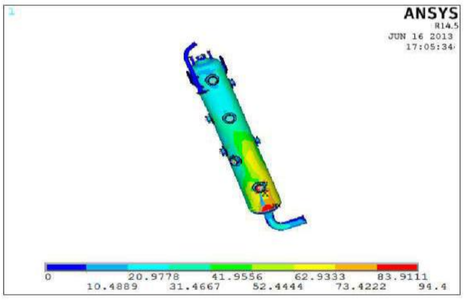

Figure 14 – Due to fluctuation of pressure – Vonmises Stress

S.N CURVE:

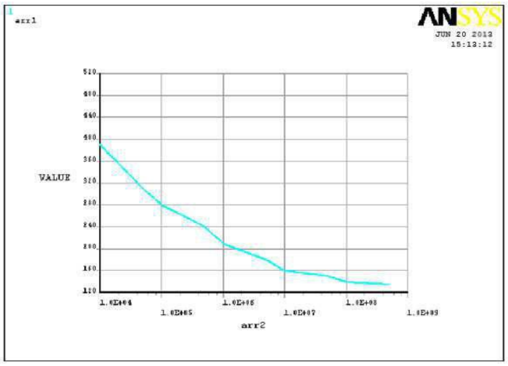

Figure 15 - SN Curve for the material

The figure 15 shows S-N curve data for the given problem. SN curve input is required for any cyclic load analysis. Stress components for both the load cases

Table 3: Stress values for both the cases

Details	Sx	Sy	Sz	Sxy	Syz	Szx
Load Case I	-1.1324	0.47490	136.01	-1.4742	15.706	-3.1026
Load Case II	-2.0851	0.91585	94.04	-1.7380	21.558	-3.0947

Fatigue Life Estimation (Corresponding maximum stress node: 958362:

- Produced Alternating SI (salt) = 30.967 Mpa
- Cycles Used/Allowed = 0.1000E+09/ 0.1000E+09
- Cumulative Fatigue Usage = 1.00000

Note: Cumulative usage factor equal to 1 represents stress generation less than the allowable endurance stress for the mater.

VI. DISCUSSION

The analysis for pressure vessel is carried out and the results are obtained using Ansys software. Due to the smaller dent region, messing can't be accommodated with three dimensional modeling. So a shell mesh is considered. Shell mesh has the advantage giving any variable property at the given height. Also Shell mesh gives faster results compared to the solid mesh. Using shell mesh concept the dent region is modeled. The dent region geometry is given 17.15mm thickness compared to the other geometrical thickness of 18 mm. RBE3 element is very useful element to apply any type of load. This gives more accurate results compared to other load transfer elements like Rbar and RBE2 elements. All 6 loads are applied using RBE3 element at the nozzle opening. The results are captured for von-mises stress. Von-mises is the most popular theory in predicting the failure of ductile materials. The results show structural safety of the problem at the dent region. The stress development is only 25Mpa. This stress is much smaller than the allowable stress of 335Mpa of the material. So structure is safe for this dent formation. Further analysis is done with three dimensional spaces. Three dimensional analyses are considered to find the effect of fillet regions at the nozzle locations. Using shell mesh, fillets can‘t be modeled. Complete structure is brick meshed using hyper mesh Boolean options to create regular geometries. Brick mesh is always better compared to the tetrahedral mesh obtained in the free mesh. Especially for graphical plots, free mesh is not suitable where as in the brick mesh; path definitions are possible where exact pattern of graph is available. The three dimensional structural analysis is done with 11 nozzle sections. Only important nozzles are considered leaving small and unimportant nozzle sections as required for the analysis. RBE3 elements are created at all nozzle sections and loads are applied as specified in the nozzle load table. The results are captured for von-mises stress. Even the thermal loads are applied as uniform loads as the difference of temperature plays for structural stress condition. The fatigue analysis shows an alternating stress development of 30.967Mpa in the problem. This stress is well below the alternating stress specified for permanent life of the pressure vessel. So structure is safe for the given loading conditions.

VII. CONCLUSIONS

The pressure vessel has been analyzed for cases of loading. In the first case a dent formation near the nozzle region is considered to find the strength and in the second case full problem is considered for analysis. The results summary is as follows.

- Initially the dent model is built using shell approach and thickness is assigned as real properties.
- Using RBE3 element, the nozzle loads are applied and results are obtained.
- The results maximum stress development of 116Mpa. This stress is less than the allowable stress of the problem.
- In the region of dent, the stress development is only 25Mpa. So the structure is safe for the given loading considerations.
- In the second case, complete assembly is analysed using Finite element analysis and results are obtained.
- The whole assembly is imported to Hyper mesh, after three dimensional modelling using Catia software. Various options in Hyper mesh are used for better quality mesh. Nozzles, welds, and shell are separated into different collectors,
- The nozzle loads are applied through RBE3 elements and pressure load is applied on the inner surface.
- Fatigue analysis is carried out for the variation of pressure load from 1.5 bars to 2.1bar with other nozzle loads constant along with the temperature variation.
- The results shows stress development of 136Mpa for 2.1 bar and 94Mpa for 1.5 bar. The stress in the weld regions shows complete safety of the problem for the given loads.
- Even fatigue analysis results shows, an alternating stress development of 30Mpa which is less than the endurance limit of 132 Mpa.
- All the results are represented through necessary pictorial views.

REFERENCES

[1] Shigley Josheph.E Mechanical Engineering Design, 2003, Sixth Edition, Mc GrawHill, Boston, Page 133, Equation 3.52-3.53, Fig 3.28.
[2] The American Society of Mechanical Engineers, “2007 ASME Boiler & Pressure Vessel Code” Section VIII Divisions 2, Alternative Rules for Construction of Pressure Vessels, Part-5, Design By Analysis Requirements.
[3] Prof. K. Lingaiah, Machine Design Data Hand Book , Suma Publishers, Second Edition, 1989.
[4] S. Ramamrutham, R. Narayan, Strength of Materials, Dhanpat Rai & Sons, 11^{th} editions ,1993.
[5] B.C. Punmia , Ashok Jain, Arun Jain, Theory of Structures – SMTS –2, Laxmi publications Pvt. Ltd, Ninth edition, 1998
[6] Norton, Robert L., Machine Design – An Integrated Approach, Prentice-Hall: New Jersey, 1998, 2 nd printing
[7] Finite Element Procedures – Klaus-Jurgen Bathe, Prentice Hall of India Pvt. Ltd.- Sixth Edition 2002
[8] Finite Element Method, by Mahadeva Siva Ramakrishna Iyer and FEA by David Heckman
[9] Finite Elements in Engineering – Tirupathi R. Chandrapatla, Ashok. D.Belegundu, rentice-Hall of India Pvt. Ltd., 2003.
[10] R. Kitching, J. K. Davise, “Limit pressures for cylindrical shells with unreinforced openings of various shapes”, Journal Mechanical
[11] R. C. Gwaltney, W. L. Greenstreet, “comparisons of theoretical and experimental stresses for spherical shells having single non-radial nozzles”, U.S Atomic Energy Commission, 1973, PP 1-33.
[12] Dennis Moss, “Pressure Vessel Design Manual”, Gulf professional publication
[13] Kang Soo kim, Suhn choi,Tae wan kim, Gyu Mahn Lee & Keun Bae Park, “Effect of opening on distance patterns & the stress distribution of the pressure vessel head”, Transactions of the 15Th International conference structural mechanical in reactor technology ,August 1999,pp 99-104.
[14] M. Giglio, “Fatigue analysis of different types of pressure vessel nozzle”, International Journal of Pressure Vessels and Piping, 2000, Vol.80, PP 1-8.
[15] You-Hong Liu, “Limit pressure and design criterion of cylindrical pressure vessels with nozzles” International Journal of Pressure Vessels and Piping, 2004, Vol. 81, PP 619–624.
[16] V.N. Skopinsky and A.B. Smetankin, “Modeling and Stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under external loading”, Int. J. of Applied Mechanics and Engineering, 2006, vol.11, No.4, PP-965-979.
[17] G.H. Rahimi, R.A. Alashti, “Lower bound to plastic load of cylinders with opening under combined loading”, Journal of Thin-Walled Structures 45 ,2007,PP- 363–370.
[18] M.Javed Hyder, M. Asif, “Optimization of location & size of opening in pressure vessel cylinder using ANSYS”, Engineering FailureAnalysis 15, 2008, pp 1-19.
[19] Ho-Sung Lee, Jong-Hoon Yoon., Jae-Sung Park” A study on failure characteristic of spherical pressure vessel”. Journal of Materials Processing Technology 164–165 (2005) 882–888.
[20] Temilade Ladokun, Farhad Nabhani* and Sara Zarei,” Accidents in Pressure Vessels: Hazard Awareness”, Proceedings of the World Congress on Engineering 2010 Vol II.
[21] Pravin Narale, “Structural Analysis of Nozzle Attachment on Pressure Vessel Design”, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue4, July-August 2012,pp.1353-1358.
[22] Avinash Kharat, Stress Concentration at Openings in Pressure Vessels – A Review”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 3, March 2013.