ESTIMATION OF STRESSES AND TEMPERATURE DISTRIBUTION AND THEIR EFFECTS IN PRESSURE VESSEL

Sampath S S, Sawan Shetty and Chithirai Pon Selvan M
School of Engineering & Information Technology, Manipal University, Dubai, UAE

ABSTRACT
This paper investigates the various stresses in a pressure vessel which are subjected to pressure. Using various thermal equations, the heat transfer taking place by conduction and convection was determined. By determining the stress concentration in a vessel, the overall performance of the system can be improved with the proper material. Further analysis is carried out to determine the deformations at various points in the vessel. An improvement in the performance is achieved by using the computational analysis software ANSYS CFX and the comparative tool DOT NET software. By considering various materials, the variation in the dimensions, stresses and heat transfer within the system are discussed.

KEYWORDS: *Pressure vessels, ANSYS CFX, DOT NET, Thermal analysis.*

I. INTRODUCTION

Thin wall pressure vessels are widely used in industry for storage and transportation of liquids and gases when configured as tanks [1]. They also appear as components of aerospace and marine vehicles such as rocket and balloon skins and submarine hulls. Two geometries are examined here namely, cylindrical pressure vessels and spherical pressure vessels. The walls of an ideal thin-wall pressure vessel act as a membrane. A sphere is the optimal geometry for a closed pressure vessel in the sense of being the most structurally efficient shape. A cylindrical vessel is somewhat less efficient for two reasons: (a) the wall stresses vary with direction, (b) closure by end caps can alter significantly the ideal membrane state, requiring additional local reinforcements [2]. However the cylindrical shape may be more convenient to fabricate and transport.

Present study involves a study of the heat transfer in the pressure vessel which is acted by the internal pressure which is generated by the fluid inside the cylinder. A different stresses which are acted within the cylinder is determined using finite element analysis technique using Ansys software [3]. Conventional or theoretical equations which govern the vessel are used and a code is generated using software called DOT NET. The stress distribution, deformation, heat transfer are calculated. Various materials are considered and the heat transfer rate and the change in the dimensions for the fixed convective coefficient value are calculated. Dot NET software gives flexibility to vary the conductivity and the optimum material selection can be performed.

In the present case a cylindrical shell with length of 1m is attached with two hemispheres at both ends, are considered. The inner diameter of the element is 0.2 m and the thickness of the cylinder is maintained 0.006 m. The pressure of the water is calculated with respect to the head of the fluid inside the cylinder. By considering the inner pressure, deformation and the stresses at various points are determined. Also by considering the temperature and the convective heat transfer coefficients, stresses and the deformations are deduced. The following element is modelled, meshed and analysed in ANSYS CFX and the results and compared. Finite element approach provides the solution at every node of consideration [4].

This paper was structured as a sequence of fundamental problems built on simple models that capture the most basic characteristics of the stress and deformation distribution in a structure. The models proceed from the simple toward the complex. The objective is to uncover the most fundamental optimization principles (or design trade-offs) that can be put to practical use in real applications. The method of analysis and optimization is the combination of heat transfer, thermodynamics and structures which is used subsequently in many engineering applications. This paper describes the methodology to determine the thermal effects and change in the dimensions. The pressure vessel was modelled, simulated and validated with the governing equations. Based on the results achieved, deformations occurring inside by the thermal effects were studied.

II. RELATED WORK

Pradeep Mani Tripathi [10] used finite element method and applied to find the temperature distribution field from the parts of cylinder head of SI engine. He studied the temperature on an engine cylinder head spark ignition, which is known constructive characteristics and the thermal parameters. The paper dealt with thermal analysis and improvement of the cylinder head assembly of SI (Spark Ignition) engine. G.R.Kannan [11] determined thermal and stress distribution on different Piston combustion chambers. Proper thermal distributions as well as mechanical stresses for different combustion chambers were considered. Analysis was carried out on four different piston head by using ANSYS (APDL 11.0) software and modelling was done with the help of 3-D Pro-e Wild fire 5.0 software.

III. METHODOLOGY

In the present study the governing equations of heat transfer between the elements are applied. Below equation (1) represents the heat transfer which takes place during conduction derived by the Fourier, and equation (2) represents the heat transfer during conduction which is by Newton's law of cooling [6]. The change in the dimensions inside the cylindrical element is shown in the equations (3), (4) & (5). The change in the dimensions [2] inside the spherical element is given by the equation (6) & (7).

Fourier's Law of conduction

$$Q^{\cdot} = -KA \frac{dT}{dx} \quad \text{.......... (1)}$$

Newton's Law of cooling

$$Q^{\cdot} = hAs\,(Ts - Ta) \quad \text{.......... (2)}$$

Change in the diameter of a cylinder due to the pressure generated inside

$$\Delta d = d[pd(2-\mu)/4tE] \quad \text{......... (3)}$$

Change in the length of a cylinder due to the pressure generated inside

$$\Delta l = l[pd(1-2\mu)/4tE] \quad \text{..........(4)}$$

Change in the volume of a cylinder due to the pressure generated inside

$$\Delta v_c = v[pd(5-4\mu)/4tE] \quad \text{.........(5)}$$

Change in the volume of a Sphere due to the pressure generated inside

$$\Delta d = d[pd(1-\mu)/4tE] \quad \text{.......... (6)}$$

Change in the volume of a sphere due to the pressure generated inside

$$\Delta v_s = v[3pd(1-\mu)/4tE] \quad \text{......... (7)}$$

Change in the volume in a pressure vessel is the sum of the change in the volumes in cylinder and sphere.

IV. MODELLING & ANALYSIS

In the present study geometric modelling of the pressure vessel is done using ANSYS CFX and its simulation is performed using the same [1]. The material of the element is acted with the pressure inside in one case and the temperature in the other. Both structural and thermal analysis is carried out. Entire set up obeys the second law of thermodynamics which requires two reservoirs to transmit the heat. One reservoir will be the inside portion of the vessel where the temperature is high. The temperature distribution will be observed from inside to the outside portion, in other words it is the low temperature reservoir. Different materials can be chosen accordingly by changing the thermal conductivity of the material. The model of the pressure vessel is shown in the figure 1.

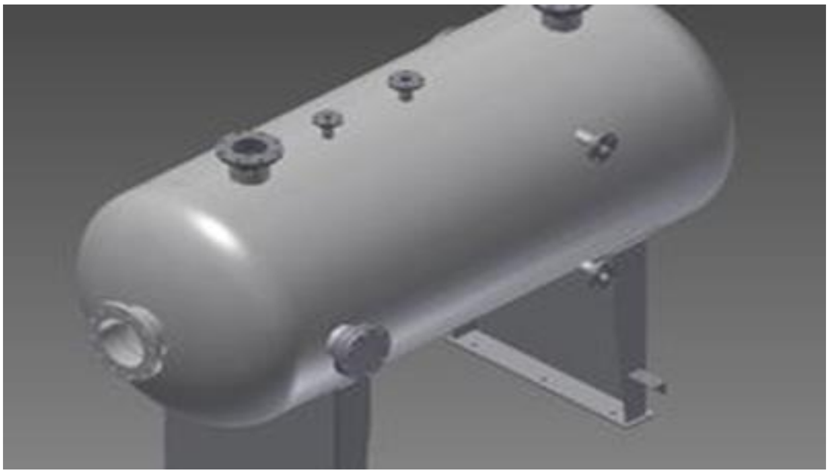

Fig 1: Model of the pressure vessel

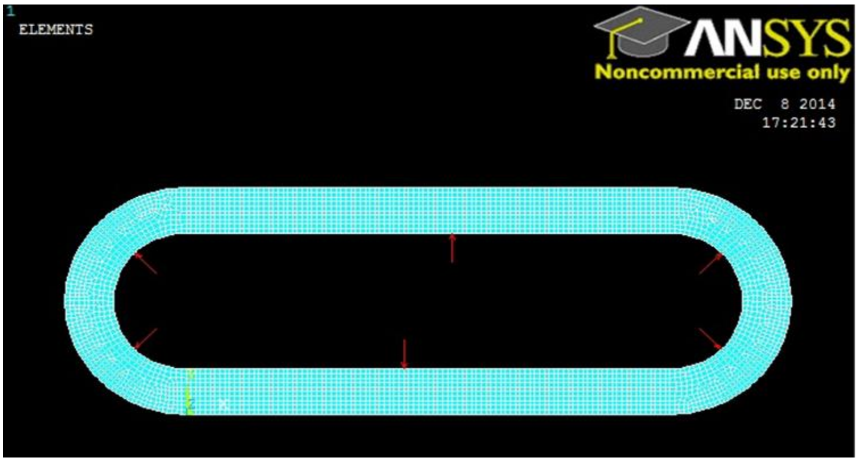

Fig 2: Meshing of the pressure vessel with the application of loads

Heating element is modeled and it is meshed using ANSYS software [9, 10]. Meshing is discretizing of an element into finite number of parts and each element is considered and solved separately. Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is oftenly used interchangeably. Typical uses are for rendering to a computer screen or for physical simulation such as finite element analysis or computational fluid dynamics [4]. After this step a structural and thermal steady state simulation is performed. By using ANSYS numerical simulation tool, whole analysis of entire assembly is performed. Present simulations adopt realistic boundary conditions by considering various different materials with different thermal conductivities. For structural analysis, pressure value of 1.5 MPa is applied inside the analysis.

Table 1: Materials and their properties

Name	Conductivity	Heat Capacity
Copper	380 W/m^0C	385 $J/Kg\ ^0C$
Construction steel Fe 360	53 W/m^0C	470 $J/Kg\ ^0C$
Aluminium Alloy EN AW-1050 A	222 W/m^0C	900 $J/Kg\ ^0C$

Due to high thermal conductivity and good machining properties, pure copper was considered as an ideal material. The second boundary in the simulation was the inner surface of the cylindrical element, where convection takes place. The value of film coefficient is assumed to be 20 $W/m2^0C$ which lies within its limits. As important boundary condition is the radiation property of the copper, due to the high film coefficient, the part of the heat flow caused by radiation is neglected in this work. Modeling and meshing is done using finite element analysis (FEA) and the simulation is performed. By means

of the numerical solution, a steady state analysis of the entire heating element is achieved. Validation of the results obtained in the FEA is carried out using Dot net.
Dot net provides user interface, data access, database connectivity, cryptography, web application development, numeric algorithms, and network communications. Governing equations are fed and the results are obtained. Classical equations parameters are varied accordingly and the output relating to this are compared with the numerical method.

V. RESULTS AND DISCUSSION

The simulation is carried out with the application of the boundary conditions by defining the pressure inside the pressure vessel. Thermal analysis is carried out by providing the thermal conductivity value to the walls and heat transfer coefficient inside the cylinder is considered. Figure 3 shows the deformation inside a pressure vessel due to the pressure acting inside.

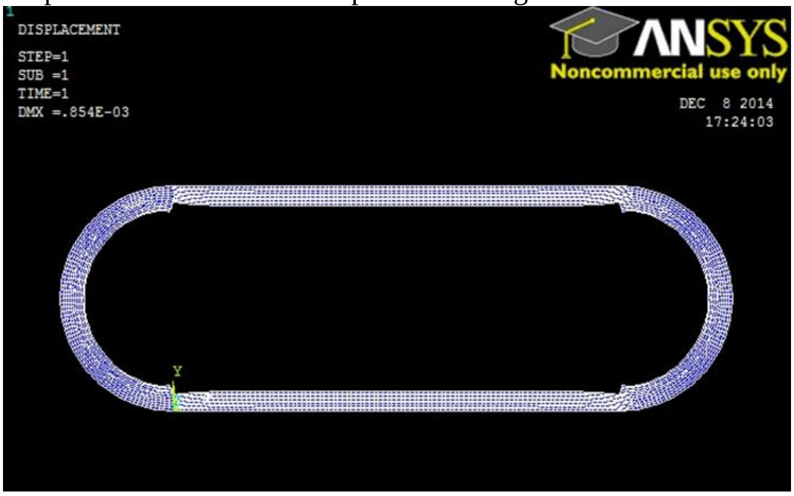

Fig 3: Deformation inside a pressure vessel due to the pressure acting inside

The above simulation shows the change in the dimensions inside the vessel due to the internal pressure. Change in the diameter, length and volume are determined using dot net. Maximum stresses are observed at the ends of the hemispheres when the radial stresses are taken into account as shown in figure 5. Also maximum stress concentration is observed at the end of the cylinder when the vessel is subjected to longitudinal stresses which are shown in figure 6. Due to the consideration of thermal conductivity for a material and the heat transfer coefficient, the temperature distribution is obtained after considering the boundary conditions.

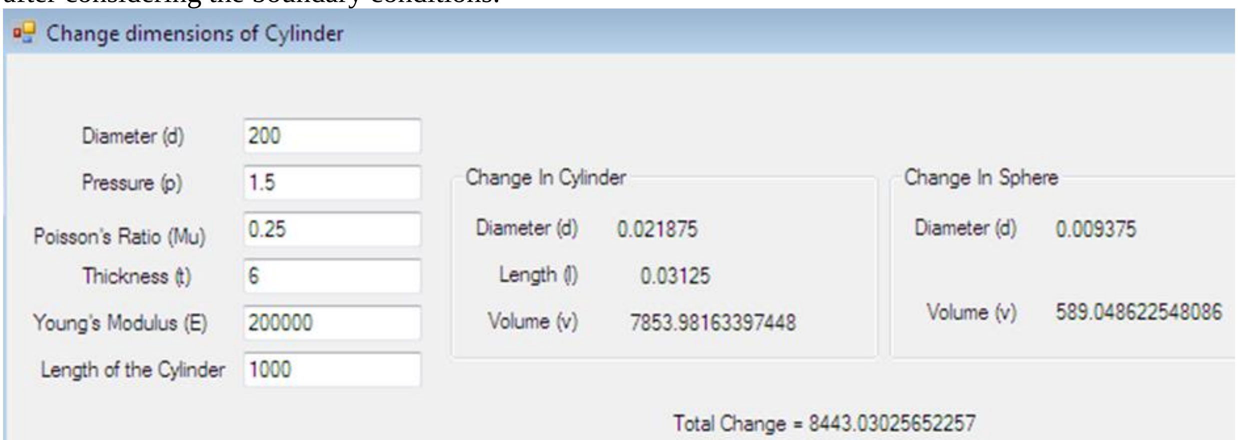

Fig 4: Change in the dimensions obtained using dot net software

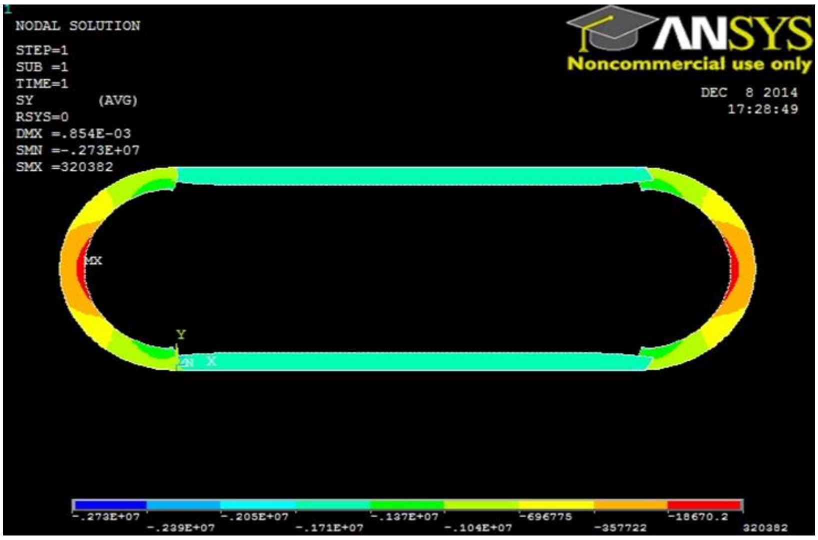

Fig 5: Stress distribution due to the consideration of radial stresses

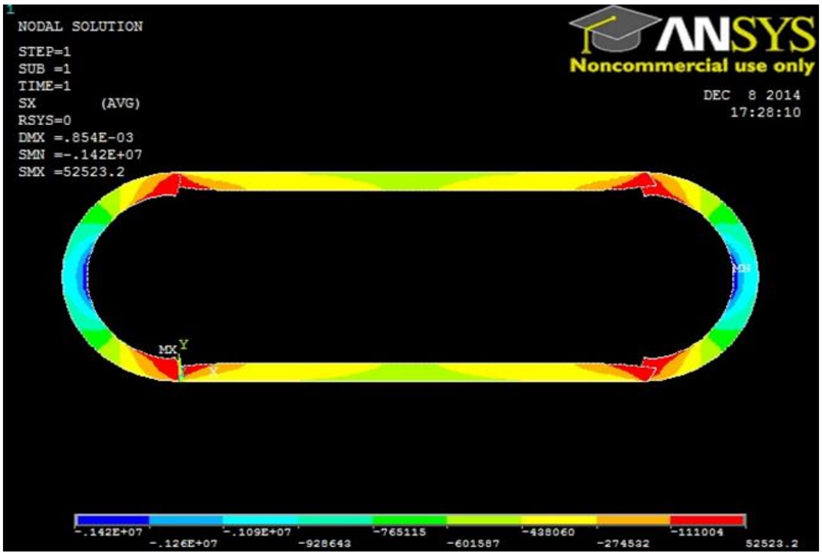

Fig 6: Stress distribution due to the consideration of longitudinal stresse

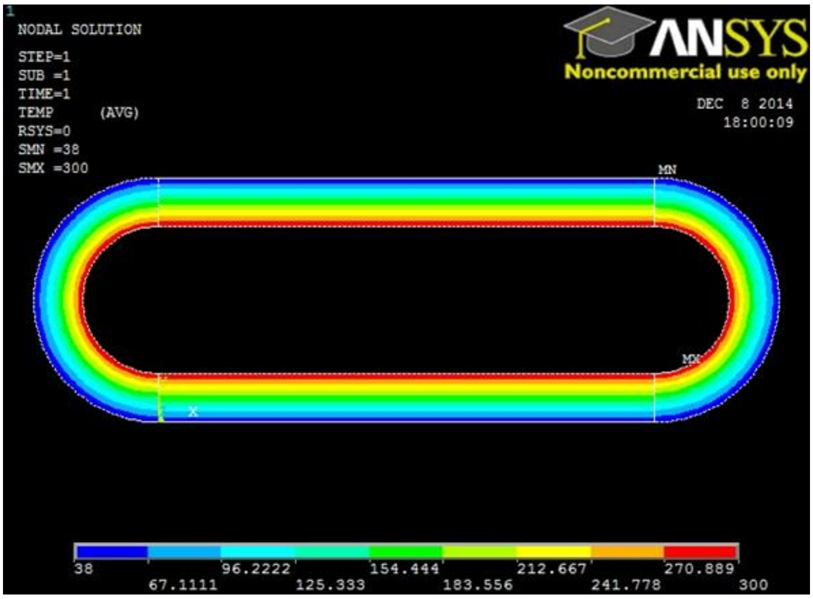

Fig 7: Temperature distribution from inside to outside the vessel

Fixed Constants: K=380 W/m^{0}C, h=20 W/m2^{0}C, thickness of the vessel=0.006 m, Internal diameter of the vessel = 0.002 m, length of the element=1 m.

Table 2: Temperature variation comparison

Thickness (from inside to outside)	Temperature distribution obtained in Finite element Analysis technique using Ansys software	Temperature distribution obtained in conventional equations using Dot Net software
0.0 mm	300^{0}C	300^{0}C
0.5 mm	277^{0}C	274^{0}C
0.1 mm	241.77^{0}C	245^{0}C
1.5 mm	230^{0}C	235^{0}C
2.0 mm	212^{0}C	216^{0}C
2.5 mm	183.55^{0}C	185^{0}C
3.0 mm	160^{0}C	163^{0}C
3.5 mm	154.44^{0}C	155^{0}C
4.0 mm	125.33^{0}C	127^{0}C
4.5 mm	96.22^{0}C	98^{0}C
5.0 mm	67.11^{0}C	70^{0}C
5.5 mm	41^{0}C	43^{0}C
6.0 mm	30^{0}C	30^{0}C

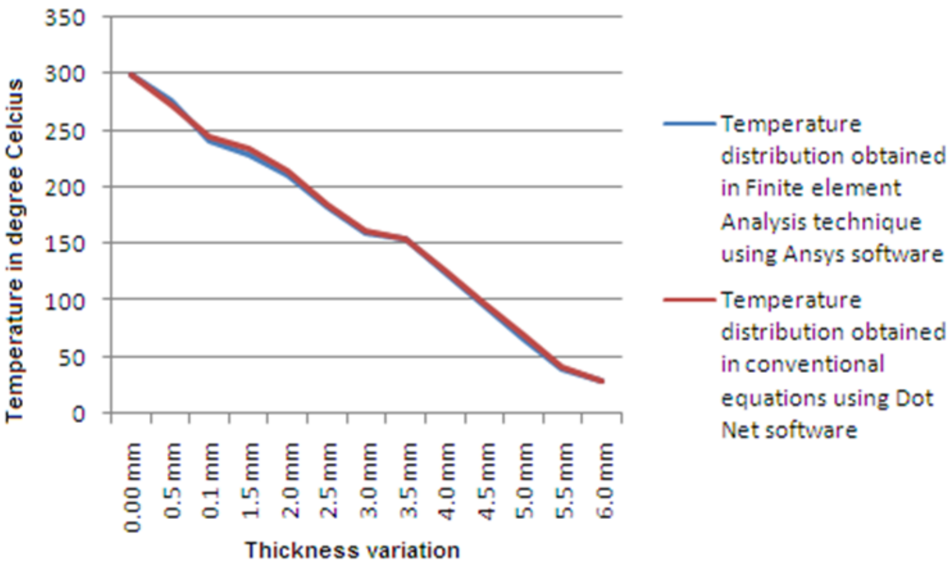

Fig 8: Comparative temperature distribution using FEA and Classical methods

Figure 8 shows the graph which represents the temperature distribution inside the pressure vessel using both classical and FEA method. Results are almost matching with each other except, there is a slight difference in the values observed in the middle.

Fixed Constants: Thickness of the Vessel=0.006 m, Internal diameter of the vessel = 0.002 m, length of the element=1 m, Poisson's ratio=0.25, Young's Modulus= 2 X 105 MPa.

Table 3: Change in the length, diameter and volume with the change in the internal pressure

Sl.No	Internal pressure (MPa)	Change in diameter of cylinder (mm)	Change in length in a cylinder (mm)	Change in volume of a cylinder (mm^3)	Change in diameter in a sphere (mm)	Change in the volume in a sphere (mm^3)	Overall change in the volume of pressure vessel (mm^3)
1	0.2	0.0029	0.0041	1047.19	0.00125	78.53	1125.73
2	0.4	0.0058	0.0083	2094.39	0.0025	157.07	2251.47
3	0.6	0.0087	0.012	3141.59	0.0037	235.61	3377.21

4	0.8	0.0116	0.0166	4188.79	0.005	314.159	4502.94
5	1.0	0.0145	0.0208	5235.98	0.00625	392.69	5628.68
6	1.2	0.0175	0.025	6283.18	0.0075	471.23	6754.42
7	1.4	0.020	0.029	7330.38	0.00875	549.77	7880.16
8	1.6	0.023	0.033	8377.58	0.01	628.31	9005.89
9	1.8	0.026	0.037	9424.77	0.0112	706.85	10131.63
10	2.0	0.0291	0.041	10471.97	0.0125	785.398	11257.35

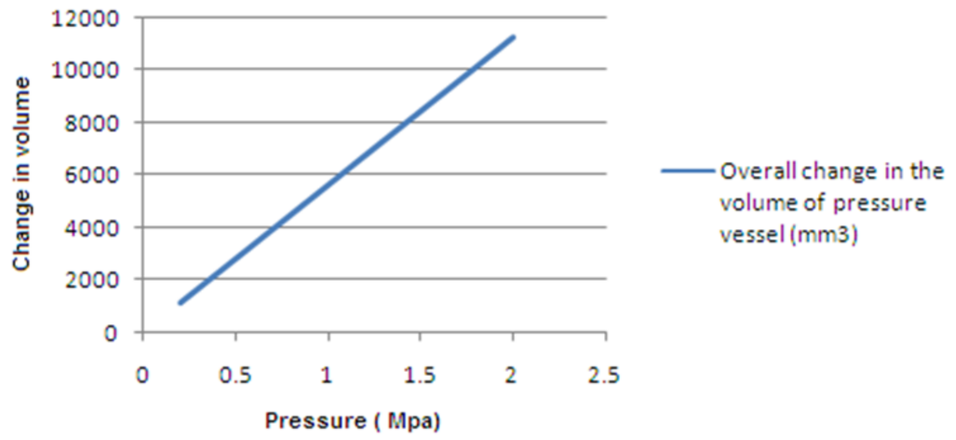

Fig 9: Variation of the volume change in the pressure vessel with the change in the internal pressure

Table 3 gives the details of change in the dimensions of pressure vessel due to the application of internal pressure. Figure 9 shows variation of the volume change in the pressure vessel with the change in the internal pressure. The graph which is plotted is a straight line, which is a linear curve.

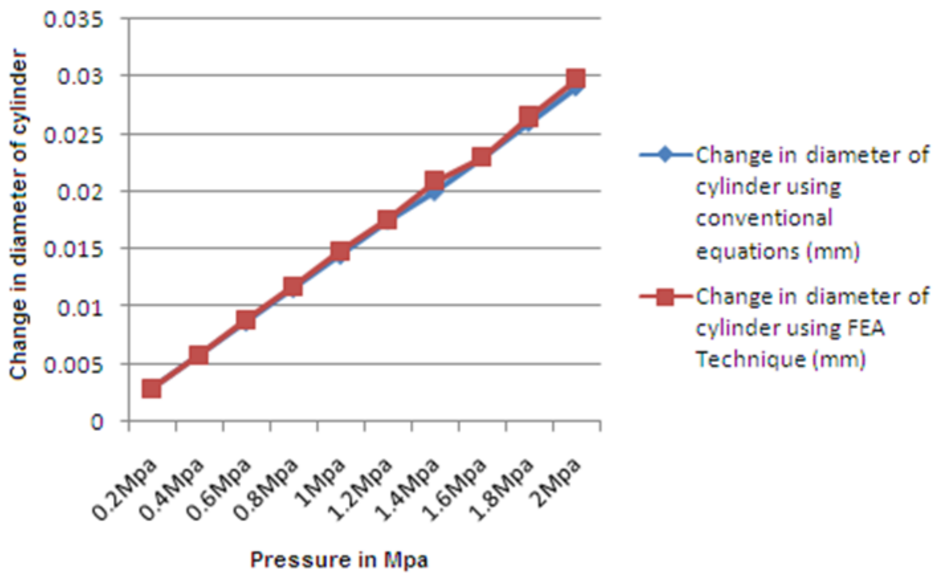

Fig 10: comparative Variation of the diametrical change in the cylinder with the change in the internal pressure

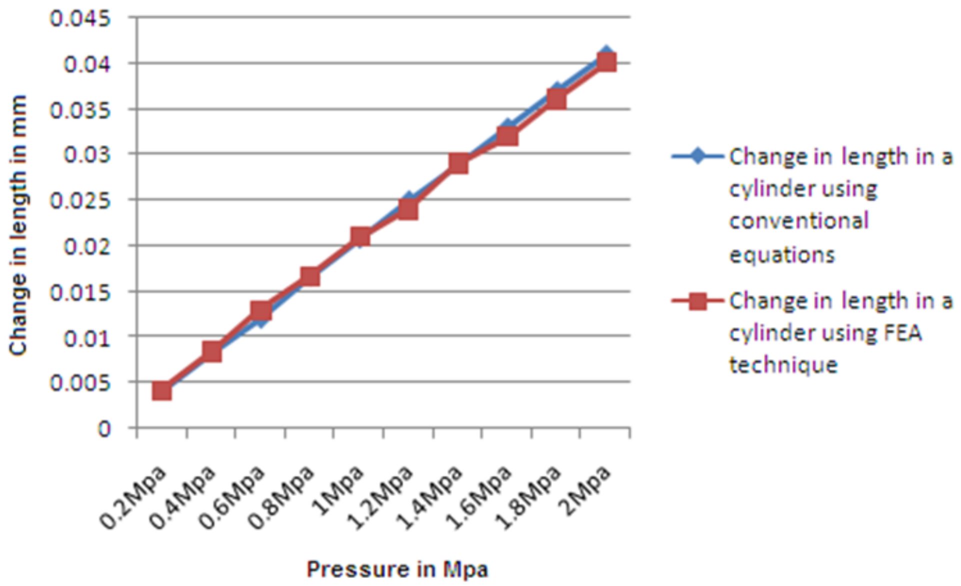

Fig 11: Comparative Variation of the change in length of the cylinder with the change in the internal pressure

Figure 10 and 11 shows the increase in change in the diameter and change in the length due to the effect of the internal pressure. The results are validated with respect to FEA technique or ANSYS software results.

VI. CONCLUSIONS

In the present analysis, a pressure vessel which is acted by the pressure at various points is investigated which enhances the temperature distribution, change in the dimensions, stresses at various points and the heat dissipation from the entire unit. An attempt is made to demonstrate the improvements to identify the maximum stress points and corresponding deformations associated with it. Deformations and other changes in the dimensions are estimated using FEA technique and the validation of this is carried out by using computer software Dot Net. It is possible to minimize stress concentration at certain points and obtain an optimum solution by selecting the material which has better thermal performances. By increasing the value of thermal conductivity it is possible to increase the heat dissipation. Modification in the shape of pressure vessel can enhance better results and improved heat transfer. Further calculations can be done by the use of dimensionless numbers like Nusselt, Prandtl, and Reynolds numbers. Due to the convection the heat transfer will be transient and the effectiveness will be increased. Analysis can be carried out by using fins in order to dissipate maximum heat into the surroundings. Thermal analysis can also be carried out by selecting composite materials. Von mises stress can be calculated for the same for the better results.

Nomenclature

Q= Heat Transfer rate, W
d= inner diameter of the pressure vessel, m
A= Cross-Sectional Area, m^2
K= Thermal Conductivity of the material, W/m^0C
h= Heat transfer coefficient, $W/m^{20}C$
T= Temperature, ^0C
A_s= Surface Area, m^2
T_s= Surface Temperature, ^0C
T_a= Ambient Temperature, ^0C
L=length of the element, m
Δ_d= change in the diameter, m
P= Pressure inside the cylinder, N/m^2
t= thickness of the wall, m
E= Young's Modulus, N/m^2

μ = Poisson's Ratio
Δl= change in the length, m
ΔV= change in the volume, m^3
V_c= Volume of the cylinder,m^3
V_s= Volume of the Sphere,m^3

REFERENCES

[1]. Sunil A. Patil, **"**Finite element analysis of optimized compound cylinder "Journal of mechanical engineering research, 2013, PP: 90-96
[2]. A. B. Ayob, M. N. Tamin, and M. Kabashi Elbasheer, "Pressure Limits of Thick-Walled Cylinders", International Multi Conference of Engineers and Computer Scientists, 2009, Vol II, PP:1-4
[3]. Harmeet Singh, Som Kumar, Nimo Singh Khundrakpam, Amandeep Singh, "Thermal Stress Analysis in Butt Welded Thick Wall Cylinder", International Journal of Engineering and Innovative Technology,2014, Vol 3, PP: 72-75
[4]. J.P.Kalejs, "Stress in thin hollow silicon cylinders grown by the edge-defined film-fed growth technique", Journal of Crystal Growth,1990,PP: 14-19
[5]. Manjunatha.T.R, Dr. Byre Gowda.H.V, Prabhunandan.G.S," Design and Static Structural Analysis of Cylinder and Piston of Two Stage Reciprocating Compressors Using ANSYS, 2013, vol 2, PP:7590-7596
[6]. Esmail M.A. Mokheimer, "Performance of annular fins with different profiles subject to variable heat transfer coefficient" International Journal of Heat and Mass Transfer, 2002, PP: 3631-3642
[7]. Pankaj N. Shrirao, Rajeshkumar U.Sambhe, Pradip R.Bodade," Convective Heat Transfer Analysis in a Circular Tube with Different Types of Internal Threads of constant Pitch", International Journal of Engineering and Advanced Technology, 2013, Vol 2, PP 335-340
[8]. Ashish Giri, Evaluation of the Performance of Annular Composite Fin using ANSYS, International Journal of Computer Applications, 2012, Vol 44, PP 4-7
[9]. Denpong Soodphakdee, Masud Behnia, and David Watabe Copeland, "A Comparison of Fin Geometries for Heat sinks in Laminar Forced Convection: Part I - Round, Elliptical, and Plate Fins in Staggered and In-Line Configurations" The International Journal of Microcircuits and Electronic Packaging, 2001, Vol 24, PP 68-76.
[10]. Pradeep Mani Tripathi, Satya Prakash, Rahul Singh, Satish Kumar Dwivedi," Thermal Analysis on Cylinder Head of SI Engine Using FEM", International Journal of Scientific Engineering and Research (IJSER),2014,Vol 2, Issue 4, PP 25-30.
[11]. G.R.Kannan , Anoop Aravind," Thermal and stress distribution of different I.C engine piston combustion chamber using 3-D finite element, "International Journal of Innovative Research in Science, Engineering and Technology"2014, Vol 3,Issue 2, PP 28-32.

AUTHOR BIOGRAPHIES

Sampath Suranjan Salins completed his Bachelors in Mechanical Engineering in 2004 from the Manipal University. He completed his Masters in Computer Aided Mechanical Design and analysis (CAMDA) in 2007 from Manipal University, Manipal. He has seven years of teaching experience. Over the years, he has taught various subjects in the field of Mechanical and Mechatronics Engineering.

Sawan Shetty completed his Bachelors in Mechanical Engineering and Master in Manufacturing Engineering from the Manipal University, Manipal. He has six years of teaching experience in the field of Mechanical and Mechatronics Engineering.

M. Chithirai Pon Selvan completed his Bachelors in Production Engineering, Masters in Computer Aided Design and Ph.D. in Mechanical Engineering. He has one and a half years industrial experience and more than sixteen years of teaching experience. He has published more than twenty research papers in international journals and has presented papers in many conferences in India, Thailand, Malaysia and United Kingdom.