Design and analysis of watertight bolted pressure chamber

MOOTAZ. E. ABO-ELNOR
Mechanical Engineering Department
Military Technical College
Elkhalifa Elmamoon St., Cairo
EGYPT
mootaz_aboelnor@hotmail.com

Abstract: - Water tightness is one of the challenges in design of submerged vessels. In this paper, a pressure proof test chamber is designed in order to test water tightness of a hatch and also for testing water prove of hull penetrators which used in submerged vessels for crossing sensor cables through the pressure hull. The test chamber is designed such that its interior simulates the wet environment (surrounding water environment) and its exterior simulates the watertight proof environment (the dry environment). This test chamber is designed to withstand internal pressure up to 20 bar; i.e. 200 meter submerge. First a three-Dimensional model of the test chamber is made using AUTODESK INVENTOR and then stress analysis is carried out using ANSYS solver provided in AUTODESK INVENTOR. Contact analysis between the pressure chamber and the assembly hatch is also carried out to examine the gap between contact surfaces of bolted region of the test chamber. Chamber shell and heads thicknesses calculated according to ASME code section-VIII Div.2 and verified through the finite element analysis. The analysis shows the ability of the test chamber to withstand the applied pressure and the effectiveness of the bolted assembly hatch.

Key-Words: **-** pressure chamber, contact analysis, assembly hatch.

I. Introduction

Pressure vessels are a commonly used device in marine engineering. Stresses applied to pressure vessels material is increases Due to the increase of the applied internal pressure in pressure vessels. Stress is the internal resistance or counterforce of a material to the distorting effects of an external force or load, which depends on the direction of applied load as well as on the plane it acts. At a given plane, there are both normal and shear stresses [1]. However, there are planes within a structural component subjected to mechanical or thermal loads that contain no shear stress. Such planes are principal planes, the directions normal to those planes are principal directions and the stresses are principal stresses. For a general three-dimensional stress state there are always three principal planes along which the principal stresses [2]. When a thin-walled cylinder, such in pressure vessel, is subjected to internal pressure, three mutually perpendicular principal stresses will be set up in the cylinder material, namely the circumferential or hoop stress, the radial stress and the longitudinal stress, [3]. Provided that the ratio of thickness to inside diameter of the cylinder is less than 1/20, it is reasonably accurate to assume that the hoop and longitudinal stresses are constant across the wall thickness, and that the magnitude of the radial stress set up is so small in comparison with the hoop and longitudinal stresses that it can be neglected. This is obviously an approximation since, in practice, it will vary from zero at the outside surface to a value equal to the internal pressure at the inside surface [4]. Until recently the primary analysis method had been hand calculations and empirical curves. New computer advances have made finite element analysis (FEA) a practical tool in the study of pressure vessels, especially in determining stresses in local areas such as penetrations, O-ring grooves and other areas difficult to analyze by regular calculation. This study set out to explore applicable methods using finite element analysis in pressure vessel analysis. One sort of pressure vessels is pressure chamber which is a closed container designed to hold gases or liquids at a pressure different from the ambient pressure. The end caps fitted to the cylindrical body are called heads and the cylindrical part of the chamber is called chamber shell.

The aim of this study is to carry out detailed design and analysis of Pressure chamber used in Testing watertight hatch and hull penetrator and to validate the using in finite element method over regular calculation methods that based on empirical formulas. The subsequence sections of this paper consists of Problem formulation which include description and design of the test pressure chamber through empirical formulas. Then a finite element method is introduce in order to study and validation of using numerical method in pressure chamber design. At the end conclusion remarks reveals the advantage of using the finite element method in analyzing chamber design especially where hull penetrators and bolted connection are introduced to the chamber.

2 Problem Formulation

2.1 Test chamber description

The test pressure chamber is made from DIN 17100 St 52-3 steel of Minimum yield strength 345 MPa. The pressure chamber is 1 meter diameter and 1.5 meter long. One end of the pressure chamber have dished end welded integrally to the cylinder. The other end is a hinged door with proper sealing (watertight hatch) integrated to bolted assembly plate as shown in Fig.1.

Fig. 1 Test pressure chamber

The hatch assembly should withstand the internal pressure during pressure prove testing. O-Ring is provided to avoid water leakage during pressure testing as shown in Fig. 2.

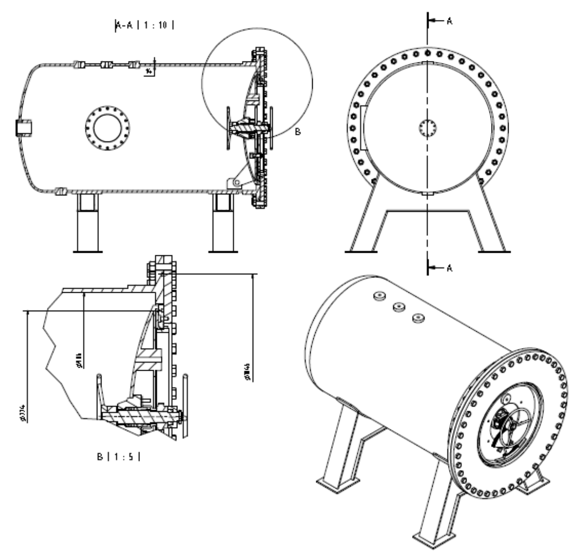

Fig. 2 Construction of test pressure chamber

2.2 Test chamber design

Two types of design methodologies are commonly applied to pressure vessels design. The most Common method is based on a simple mechanics approach and is applicable to “thin wall” Pressure vessels which by definition have a ratio of inner radius (R) to wall thickness (t) of R/t≥10. The second method is based on elasticity solution and is always applicable regardless the R/t ratio and can be referred to as the solution for “thick wall” pressure vessels [5]. First type of design methodology and analysis is discussed here, although for most engineering applications, the thin wall pressure vessel can be used. Pressure vessels design using ASME code section-VIII Div.1 is examined [6]. Comparison between ASME code section-VIII Div.1 and ASME code section-VIII Div.2 shows the ability of Div.2 of considering combined stress over Div.1 [7, 8]. In this study the chamber analysis is carried out According to ASME code section-VIII Div.2.

According to ASME code section-VIII Div.2 and thin-wall shell design, the following relations are used

Shell thickness can be calculated as

$$t = \frac{PR}{SE-0.6P} \tag{1}$$

$$t \geq \frac{290*17.95}{35534.24*1-0.6*290} \geq 0.147\ in$$

I.e. t ≥ 4 mm.

The test chamber was designed with 4 safety factor, then t=16 mm

Ellipsoidal Head thickness can be calculated as

$$t = \frac{PLM}{2SE-0.2P} \tag{2}$$

$$t \geq \frac{290*39.6*1.52}{2*35534.24*1-0.2*290} \geq 0.245\ in$$

t ≥ 6.2mm, the test chamber Ellipsoidal Head thickness is 16 mm, i.e. with safety factor 2.5. Table 1 conclude used parameters for solving equation (1) and equation (2)

Table 1. ASME code section-VIII Div.2 Parameters

NOTATION	MKS		SI	
P.. Internal pressure (Psi)	2	MPa	290	Psi
R.. Shell radius (in)	456	mm	17.95	in
S.. Allowable stress (Psi)	245	MPa	35534.24	Psi
E.. Joint efficiency	1		1	
L.. Ellipsoidal Head inside crown radius (in)	1006	mm	39.6	in
M.. factor depending on head geometry	1.52		1.52	

2.3 Assembly hatch bolt calculation based on ASME code

Bolted flange joints perform a very important structural role in the closure of flanges in a pressure vessel. It has two important functions: (a) to maintain the structural integrity of the joint itself, and (b) to prevent leakage through the gasket which is already preloaded by bolts [9]. In this chamber design, one flange is having a groove for gasket installation, and other flange is flat. The two flanges are connected by a number of bolts/studs. Assembly bolts preload is extremely important for the successful performance of the joint. The preload must be sufficiently large to seat the gasket and at the same time not excessive enough to crush it. The flange stiffness in conjunction with the bolt preload provides the necessary surface and the compressive force to prevent leakage of the water contained in the pressure chamber. As the pressure of the water inside the chamber increases, bolt preload decreases, which reduces gasket compression and tends to separate the flange faces and then water leakage is introduce and water tightness fails. In full sealed joints uniformity of pressure on the sealing element is important. To maintain adequate uniformity of pressure; adjacent bolts should not be placed more than six nominal diameters apart on the bolt circle. To maintain wrench clearance, bolts should be placed at least three diameters apart [10].

A rough rule for bolt spacing around a bolt circle is such that

$$6Nd \geq \pi D_b \geq 3\text{Nd} \tag{3}$$

Where: N is the number of bolts = 40 bolt

D_b is the diameter of bolt circle =1100 mm

D bolt diameter which calculated as

i.e. $14.39 \leq d \leq 28.7$ mm

Bolt diameter is selected to be d=24 mm with 60 mm active length. The tight torque of M24 bolt is T=1150N.m

Considering torque coefficient K=0.25.

Bolt installation force can be calculated as

$$F_{install} = \frac{T}{K*d} \tag{4}$$

$$F_{install} = \frac{1150000}{0.25*24} = 191.6\ kN$$

For the test chamber, the area affected by 2MPa pressure (20 bar) of the bolted flange is of diameter of 912 mm. So total pressure force affecting the bolted flange is 1305838 N. Each bolt of the assembly bolts should support

$$F_{load} = 1305838/40 = 32.6459\text{ kN}.$$

Total force applied to the bolt is

$$F_{bolt} = F_{load} + F_{install} \tag{5}$$

$$F_{bolt} = 32.6459 + 191.6 = 224.3125\ kN$$

Bolt stress

$$\sigma = \frac{F_{bolt}}{A_{bolt}} \quad (6)$$

$$\sigma = \frac{224312.5*4}{\pi*(24)^2} = 496\ MPa$$

I.e. bolt strength should be more than 496 MPa. The selected bolt is of grade 10.9 with minimum proof strength of 1060 MPa [11].

Bolt elongation due to installation force

$$\delta_1 = \frac{F.L}{E.A} \quad (7)$$

$$\delta_1 = \frac{239500*60*4}{207000*\pi*(24)^2} = 0.15\text{ mm}$$

Bolt elongation due to 2MPa pressure

$$\delta_2 = \frac{F.L}{E.A} \quad (8)$$

$$\delta_2 = \frac{32645.9*60*4}{207000*\pi*(24)^2} = 0.0209\text{ mm}$$

Total Elongation of the M24 bolt

$$\delta = \delta_1 + \delta_2 \quad (9)$$

$$\delta = 0.15 + 0.0209 = 0.175\text{ mm}$$

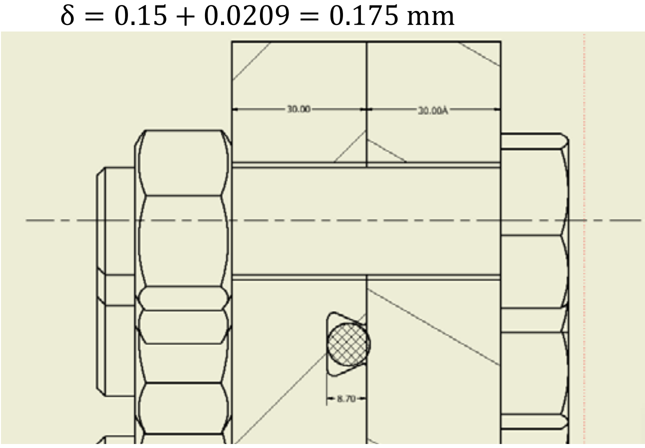

Fig. 3 Bolted hatch with O-Ring assembly

For the 9.52 mm O-ring used in this design, the actual squeeze of installation, according to Parker [12], is 9.52-8.7= 0.82mm as shown in Fig. 3, which means that the bolt elongation due to the 2MPa load (0.0209 mm) will not affect the O-ring sealing

2.4 Finite element modeling and analysis

To validate the design calculations using ASME code section-VIII Div.2 in the previous sections, a finite element analysis was carried out using ANSYS solver with 506208 3D elements, as shown in Fig. 4.

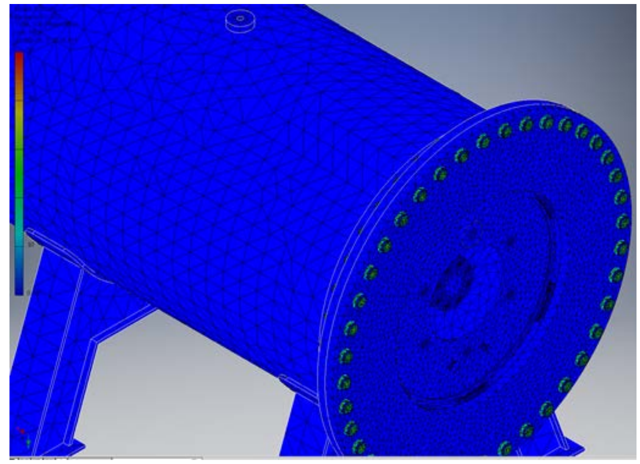
Fig. 4 3D finite element model mesh

Analysis Boundary conditions is set such that; Fig. 5:

- One support of the four support legs of the test chamber is fixed, the other three supports are set as frictionless contact with the ground to enable real simulation of the chamber elongation when subjected to internal pressure.

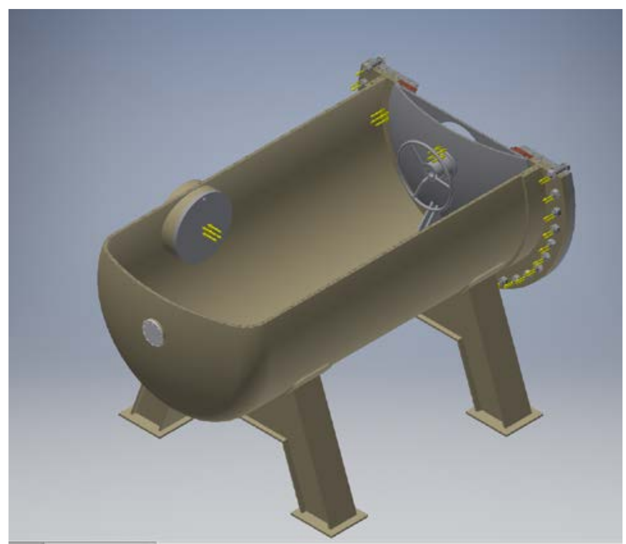
Fig. 5 Applied forces to the 3D model

- Contact parts are consider to be even bonded or frictionless depends on its operation condition.
- To enable bolt preload elongation and elongation due to applied pressure, each bolt and nut contact is set as frictionless contact.
- Each bolt flange head is bonded to the assembly hatch flange.

Bolted joints have been simulated using finite element method in the literature [13, 14]. In Fig. 5 internal pressure is applied to the chamber cylindrical shell, welded dished end (elliptical head), assembly hatch and all covers of hull penetrator openings.

To establish initial contact of the assembly hatch in the test chamber; bolt preload force is applied at both bolts and nuts as shown in Fig. 6. This preload force is the install force calculated using equation (4) and is set as 191.6 kN for the used M24 bolt.

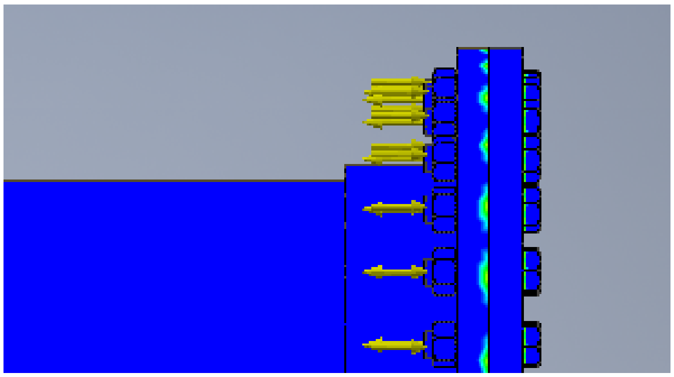

Fig. 6 Bolts preload

This preload force is physically introduced during bolt assembly and it should be related to bolt tighten torque.

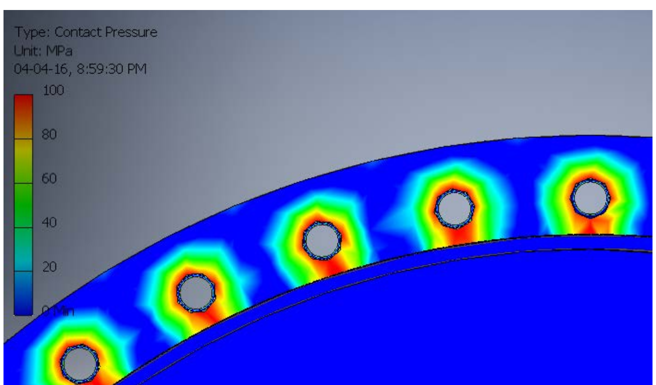

Fig. 7 Contact pressure of bolted region

Fig. 7 shows contact pressure region applied to the assembly hatch flange due to the preload of the assembly bolts. Similar contact region was obtained in the literature [15].

3 Problem Solution

To study the effect of chamber internal pressure on chamber shell, elliptical head, bolts stresses and contact pressure of the assembly hatch; a 2MPa pressure was applied to the chamber interior surfaces.

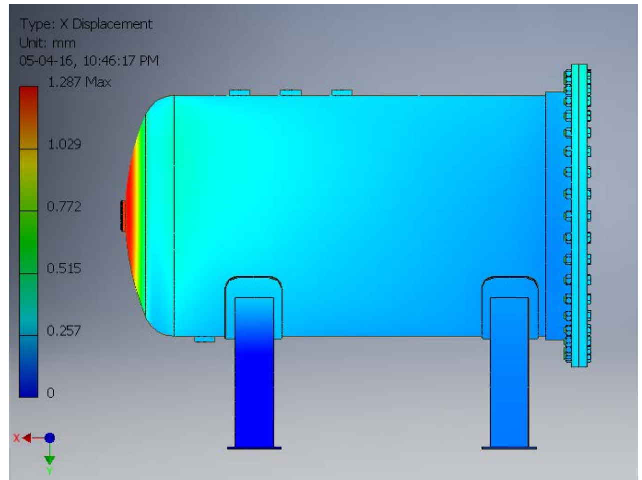

Fig. 8 Chamber axial displacement

Fig. 8 represents axial displacement of the test chamber, the homogenous elongation (displacement) of the chamber shows the adequate placement of the boundary condition.

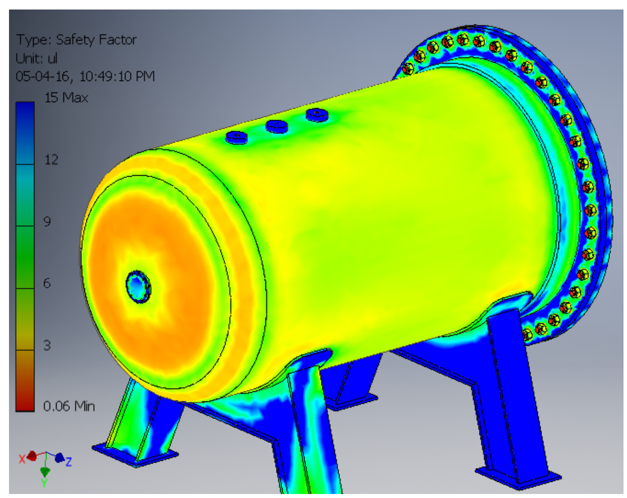

Fig. 9 Chamber safety factor

Fig. 9 represents the overall safety factor of the test chamber which found to be in the order of 4 for the cylindrical shell and more than 2 for the ellipsoidal head (dished end). Values of safety factor obtained analytically based on ASME code section-VIII Div.2 are in the same order of those obtained using the finite element modeling. The thickness of the ellipsoidal head (dished end) should be more than the thickness of the cylindrical shell to ensure equal distribution of stress/strength or safety factor.

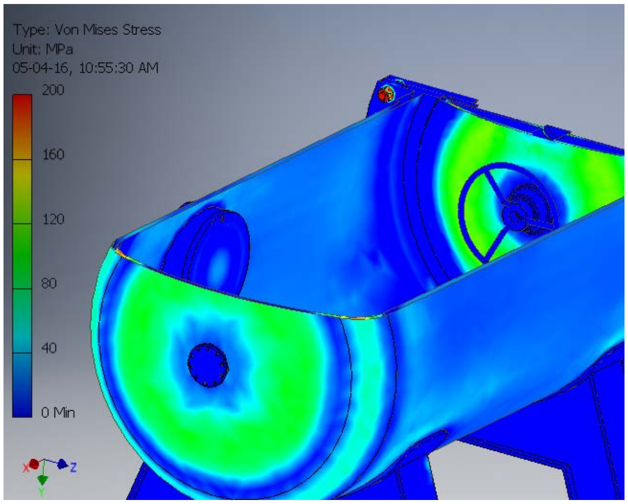

Fig. 10 Dish-end stresses and Hatch stresses

Fig. 10 represents Von Mises stresses due to applying of 2 MPa internal pressure. The figures show that the dish-end and hatch bow are highly affected with the applied pressure, also it is cleared from Fig. 10 that the dish-end stress is in the order of 200 MPa, and hence the DIN 17100 St 52-3 steel is safe to produce such chamber.

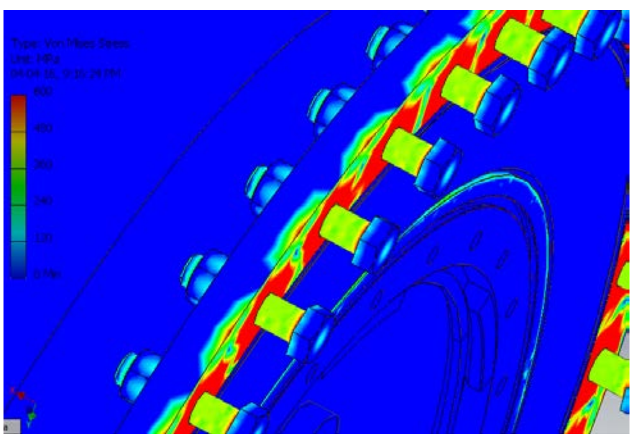
Fig. 11 Bolts stresses

To study the effect of the applied pressure on the bolts of the bolted assembly hatch region, the assembly hatch was hidden in Fig. 11 and bolts stresses is represented in the figure, the finite element analysis shows bolts stresses in the order of 500 MPa which matches the analytical calculations (496 MPa). Fig. 11 also shows stress distribution on the contact region of the bolted assembly hatch. Contact pressure of the contact region is analyzed using finite element method and is blotted in Fig. 12, which shows contact pressure distribution in the contact region of the bolted assembly hatch flange.
Comparing Fig. 7 and Fig. 12 shows that contact pressure is reduced after applying 2MPa pressure which indicate the reduction of the contact pressure due to bolt elongation.

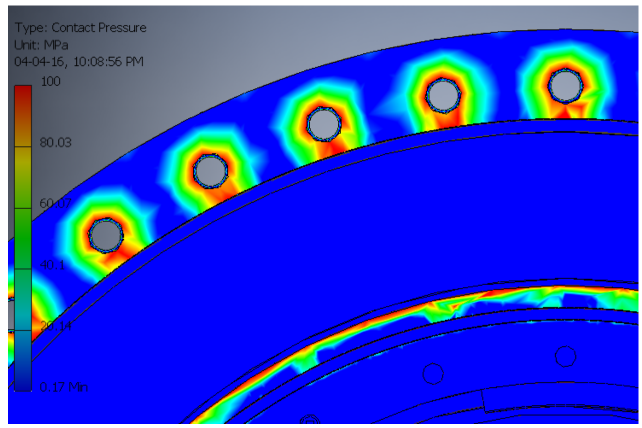

Fig. 12 Contact pressure of bolted region

4 Conclusion

Test pressure chamber can be designed based on ASME code section-VIII Div.2. Using finite element method shows that Dish-end, penetrators and assembly hatch are the most critical parts in the design of the test chamber where stresses are concentrated in these regions. Dish-end thickness should be more than cylindrical shell thickness. Bolt selection should be considered based on both stress and elongation. Bolt preload is extremely important to establish sufficient contact pressure of the contact region and compensates reduction of contact pressure after applying pressure in the chamber. The design is found to be satisfactory since result obtained from analytical calculations is validated by the finite element analysis besides the advantage of using finite element method in analysis critical regions such in hull penetrators and bolted connection of the assembly hatch.

References

[1] Design for Manufacturability Reference and Training Book. Stress-Strength of Materials, Engineers Edge (2010)

[2] Spence. J & Tooth. A.S. (1994). Pressure Vessel Design: Concepts and Principles. Taylor & Francis; 1st edition, London

[3] Sharma. S. C. 2010. Strength of Materials. Department of Mechanical & Industrial Engineering. Indian Institute of Technology Roorkee. URL: http://nptel.iitm.ac.in/courses/Webcourse.

[4] Hearn.E.J., Mechanics of Materials, 2nd edition, Cambridge University Press, United Kingdom, 1998.

[5] Vishal V. Saidpatil, Arun S. Thakare, “Design & Weight Optimization of Pressure Vessel Due to ThicknessUsing Finite Element Analysis”, International Journal of Emerging Engineering Research and Technology, Volume 2, Issue 3, June 2014, PP 1-8

[6] B.S.Thakkar, S.A.Thakkar, “DESIGN OF PRESSURE VESSEL USING ASME CODE, SECTION VIII, DIVISION 1”, International Journal of Advanced Engineering Research and Studies IJAERS, Vol. I, Issue II, January-March, 2012/228-234

[7] Apurva R. Pendbhaje, Mahesh Gaikwad et al, “ DESIGN AND ANALYSIS OF PRESSURE VESSEL”, International Journal of Innovative Research in Technology & Science(IJIRTS), VOLUME 2, NUMBER 3, may 2014, (Page no:28-34)

[8] ASME Code, Section VIII, Division 2

[9] Nomesh Kumar, P.V.G. Brahamanandam and B.V. Papa Rao, “3-D Finite Element Analysis of Bolted Flange Joint of Pressure Vessel”, MIT International Journal of Mechanical Engineering Vol. 1, No. 1, Jan 2011, pp 35-40

[10] Shigley, J. E. and Mischke, C. R., Mechanical Engineering Design, tenth Edition, McGraw-Hill, Inc., New York, 2014.

[11] Mechanical properties of fasteners made of carbon steel and alloy steel, ISO 898-1:2013.

[12] Parker O-Ring Handbook ORD 5700, 2015.

[13] Khemchand M. Kapgate, V. D. Dhopte, “Techniques for Design of Bolted Joint in Finite Element Analysis”, IJSTE - International Journal of Science Technology & Engineering, Volume 2, Issue 2, August 2015.

[14] Jerome Montgomery, “Method for modelling bolt in bolted joint”, Siemens Westinghouse Power Corporation, Orlando, FL.

[15] Raj Kiran Ilony and A.Purushotham, “Design and Analysis of Canister Testing Chamber”, International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.6, Nov-Dec. 2012 pp-4443-4449