IJE TRANSACTIONS A: Basics Vol. 31, No. 1, (January 2018) 58-64

International Journal of Engineering

Journal Homepage: www.ije.ir

TECHNICAL NOTE

Design and Analysis of Pressure Vessel Subjected to Pressure-temperature Variation

S. Parkhe, Annamalai K.*

School of Mechanical and Building Sciences, VIT University, Chennai, Tamilnadu, India

PAPER INFO

Paper history:
Received 24 April 2017
Received in revised form 20 September 2017
Accepted 30 November 2017

Keywords:
Pressure Vessel
Finite Element Method
Fatigue Analysis
Structural Analysis
Thermal Analysis
Fatigue Life

ABSTRACT

Pressure vessel is a container used to store fluids under pressure and temperature. The fluids can be air, water, chemicals, fuel, gases etc. are most commonly used in food and chemical industries, oil refineries and so on. Pressure vessel is subjected to thermal and structural loads for power plant applications. Since the pressure vessel are subjected to both structural and thermal loads stresses, the design of pressure vessel was done using standard code and design methods. In this paper a design methodology is developed for pressure vessel used in Coker Blow-down application based on ASME Section VIII, Division 1 design code. The design methodology has been developed and the same is verified with numerical method so that it will not fail in case of variable pressure and temperature condition. The modelling has been done using SolidWorks-2015 and analysis is done using ANSYS.

***doi**: 10.5829/ije.2018.31.01a.09*

1. INTRODUCTION

Pressure vessel is a container used to store fluids which are under pressure and temperature. Pressure vessel is design to withstand both thermal and structural stresses. This pressure vessel is subjected to cyclic operation conditions. In steam out to Fractionator operation the mixture of highly vaporized steam is separated with the help of Fractionator. The Fractionator separates the highly vaporized mixture of steam by vapour to cool, condense and vapour again. In steam out to Blow down operation the concentrated impurities are removed from the steam. This operation is also used for removing undesired solids or foreign particles from the steam. In Decoking operation, the fluid is fed into Coker and heated to cracking temperature. The Coke drum then separates lighter vapours from fluid including hydro carbon gases and heavy gas particles. First, during steaming and cooling of each Coke Drum, the column receives steam/Hydro Carbon vapour with feed temperature ranging from 441^{0}C at the beginning to 177 ^{0}C at the end of the 7 hour period. Then, the column is idle until the next decoking cycle. Also, during the coke drum warm up (Approx. 1.5 hrs. period) the column receives a two phase Hydro Carbon feed at temperatures ranging from 121^{0}C to 371^{0}C. No ladder / platform & pipe support clips shall be welded on this column. In this paper work we are working on structural and thermal analysis of pressure vessel.

The detailed design and analysis of pressure vessel needs to be done for optimum thickness, required temperature distribution and dynamic behaviour withstand [1]. Finite Element Analysis is used to determine the discontinuity stresses in misaligned circumferential joint of cylindrical pressure vessels. Also, peak stress values obtained from the design formulae have been verified with finite element method (FEA) [2]. The hemispherical head pressure vessel has low stress distributed as compared to other heads. For most applications, elliptical head is selected. An equal stress distribution criterion in hemispherical head hence for higher pressure application hemispherical head has been followed [3]. The induced stress calculations using Ansys coupled field analysis for thermo-mechanical loading. Primary stresses are due to inside pressure and secondary stresses are due to thermal loading. FEA results are within 15% error limit when compared to analytical results [4]. The pressure vessel is analysed forthermal loads, pressure loads and combined thermal

*Corresponding Author's Email: *kannamalai@vit.ac.in* (Annamalai K.)

Please cite this article as: S. Parkhe, Annamalai K., Design and Analysis of Pressure Vessel Subjected to Pressure-temperature Variation, International Journal of Engineering (IJE), IJE TRANSACTIONS A: Basics Vol. 31, No. 1, (January 2018) 58-64

and pressure loads are also analysed for induced stresses to show that the developed stresses and temperature are within controlled values [5]. When openings are provided on the walls of pressure vessel, they result in the pressure restraining boundary and hence cause a discontinuity, and as a result, stress concentration can occur. They found that stress concentration factor increases by increasing the opening size in geometry [6]. In case of vertical expansion of vessel either due to thermal expansion or due to vertical loading, the legs are susceptible to buckling. The quantities of stress and deformation are safe for all values of the increment in the angle of leg support [7]. In the pressure vessel, ribs (stiffeners) are provided around the shell part of the vessel to avoid the buckling failure. The stiffeners provide maximum strength in case of buckling and bending failure of pressure vessel [8].

In this paper a design methodology is developed for design of pressure vessel used in Coker Blow-down application based on ASME Section VIII, Division 1 design code. The developed design methodology has been verified with numerical method using finite element analysis so that design validation is checked in case of variable pressure and temperature conditions. The modelling has been done using SolidWorks-2015 and FEA analysis is done using ANSYS.

2. METHODOLOGY

This pressure vessel is a vertical column and it is rested on the skirt support vertically. In this pressure vessel, the main parameters of the working fluid/substance i.e. pressure and temperature are not constant. The pressure and temperature are varying continuously throughout the complete operation of the vessel from 144 $^{\circ}$C to 441 $^{\circ}$ C.

As the pressure and temperature are variable during the operation of the pressure vessel, finite element analysis is carried out to study the stress generation in each process of operation and to find out the fatigue life of the pressure vessel.

A. Design Model Using Pv-Elite 2016 Software:-
The design of pressure vessel is carried out using PV-Elite 2016 software. The PV-Elite 2016 software provides the minimum required thickness dimensions of the pressure vessel based on the material used for the pressure vessel, internal and external pressure and temperature applied on the pressure vessel, etc,, as shown in Figure 1. Also, the PV-Elite 2016 software provides the dimensions for the support equipment which require withstanding the pressure vessel assembly.

B. Modelling Using Solidworks-2015:-
3-D modelling is carried out in Solid Works – 2015 as per the design data provided by the client as shown in Figure 2. Inner geometrical details are shown in Figure 3. The design data of Pressure Vessel is as per the ASME section –VIII, Division – 1.

C. Finite Element Analysis
The STEP file is imported in the Ansys – 14.5 Workbench. The engineering data is provided as per the requirement of Pressure vessel. After the pre-processing, the meshing of pressure vessel is carried out. The element type of meshing are Solid 186 and Solid 187, number of elements are 376819.

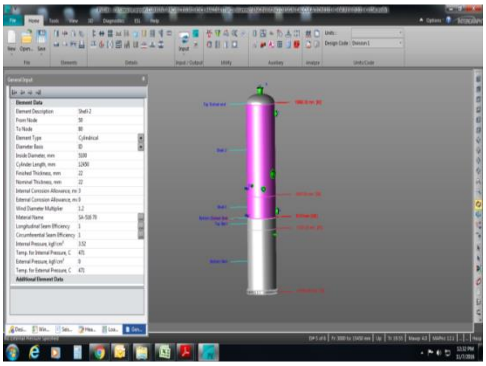

Figure 1. PV-Elite Software Model

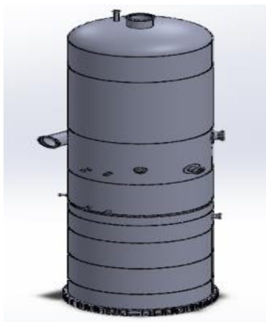

Figure 2. 3-D Model of Geometry

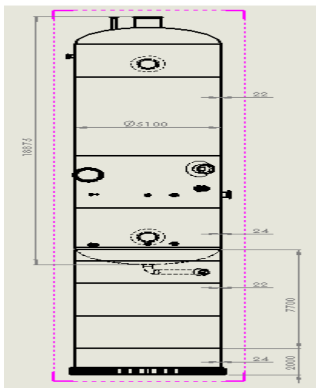

Figure 3. Thickness and Height of Vessel (Draft view)

The meshed model is shown in Figure 4. After meshing, the boundary conditions are applied as per the provided design data. The boundary conditions are Top vessel internal temperature, Bottom vessel internal temperature, Top vessel internal pressure, Bottom vessel internal pressure, Thrust on nozzles, Piping loads on nozzles, Convection, Fire proofing on Skirt of the vessel etc. The piping loads are not applicable for the nozzles which are manholes, vent/exhauster, level transmitters, etc. Figures 5 - 12 show the relevent boundary conditions applied on the model.

In the FEA of pressure vessel, the thermal and structural analysis is carried out to find out maximum stress values in the pressure vessel. By using the stress values, it is possible to do fatigue analysis of the vessel.

D.Material Properties For Analysis:-

TABLE 1. Material Properties

Material Properties					
Material	Design Temp. (°C)	Modulus Elasticity (MPa)	Yield Strength (MPa)	Density (kg/m^3)	Poisson Ratio
SA 516 Gr. 70	471	157.38E3	164.64	7750	0.3
SA 106 Gr. B	471	157.38E3	154.64	7750	0.3
SA 105	471	157.38E3	158.64	7750	0.3

E. Process/Operations In Pressure Vessel:-

TABLE 2. Process/Operations in Pressure Vessel

Process/Operation	Temp. (°C) Top/Outlet	Temp. (°C) Bottom/Inlet	Pressure (kg/cm^2 g)
Process Start	40		
Steam out to Fractionator	40	177	0.56
Steam out to Blowdown	177	441	0.77
Quenching and Filling	121	343	0.77
Water Drain and Unhead	40	177	0.56
Decoking Operation	40	177	0.56
Rehead and Pressure Test	40	177	0.56
Preheat	40	177	0.56

F. Meshing Of Pressure Vessel:-

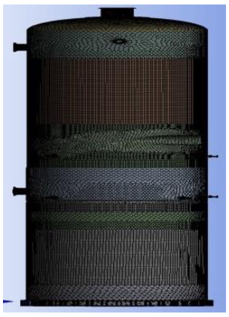

Figure 4. Meshing of the Pressure Vessel

G. Boundary Conditions:-

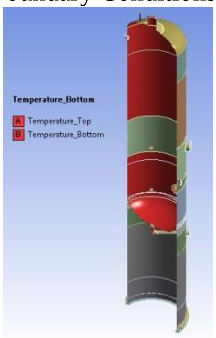

Figure 5 Internal Temperature

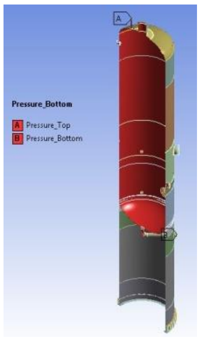

Figure 6. Internal Pressure

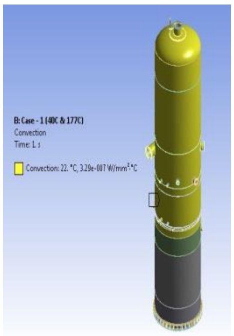

Figure 7. Convection

Figure 8. Fire Proofing for Skirt of Vessel

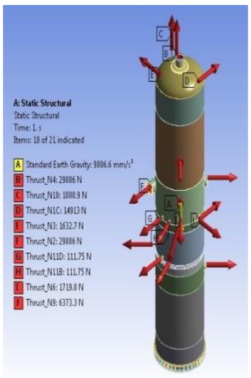

Figure 9. Thrust on Nozzle

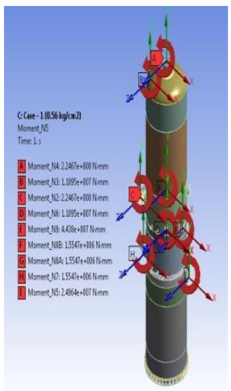

Figure 10. Piping loads

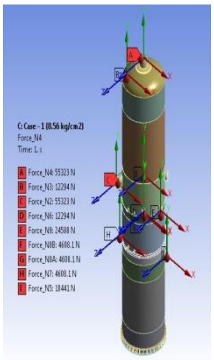

Figure 11. Piping loads on Nozzle (Force)

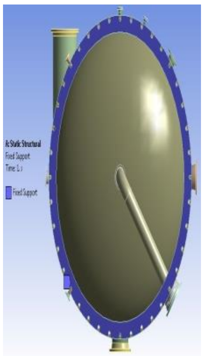

Figure 12. Fix Support

3. RESULTS

A. Structural And Thermal Analysis Cases:-

TABLE 3. Structural and Thermal Analysis Cases

Cases	Process/Operation	Temp. (°C) Top/Outlet	Temp. (°C) Bottom/Inlet	Pressure (kg/cm^2 g)
I	Steam out to Fractionator	40	177	0.56
II	Steam out to Blowdown	177	441	0.77
III	Quenching and Filling	121	343	0.77
IV	Water Drain and Unhead	40	177	0.56

B. Structural And Thermal Analysis Results For Original Geometry:-

The FEA results from Figures 13 - 16 show maximum stress results of the original geometry with reference to Figure 18.

C. Modification In The Geometry:-

In the above analysis from Figures 13 - 16, it is found that even though the nozzles of pressure vessel are safe, but the stress values are slightly more around the nozzle.

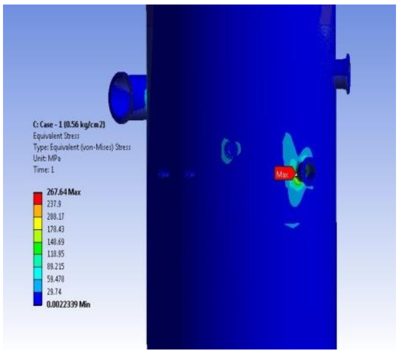

Figure 13. Maximum Stress Location (Case – I)

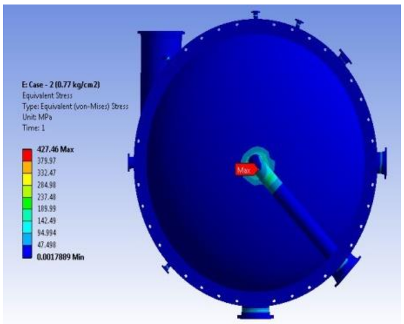

Figure 14. Maximum Stress Location (Case – II)

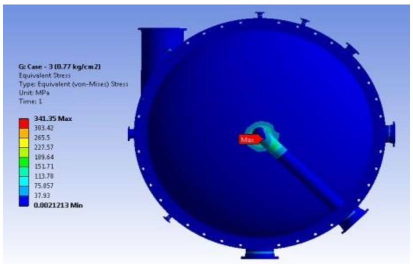

Figure 15. Maximum Stress Location (Case – III)

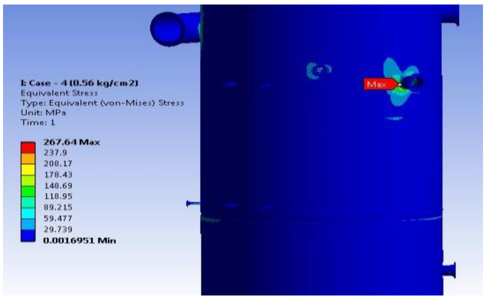

Figure 16 Maximum Stress Location (Case – IV)

Hence, to reduce the value of stress concentration, an insert plate is added around the nozzle instead of reinforcement pad and the analysis of the modified geometry is carried out to compare the fatigue life of the vessel.

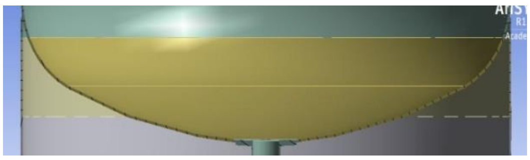

Figure 17. Modified Geometry

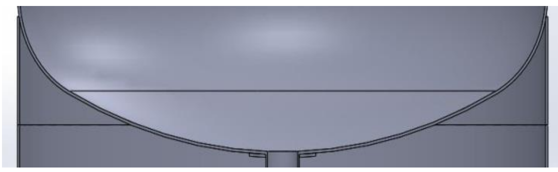

Figure 18. Original Geometry

D. Structural And Thermal Analysis Results For Modified Geometry:-

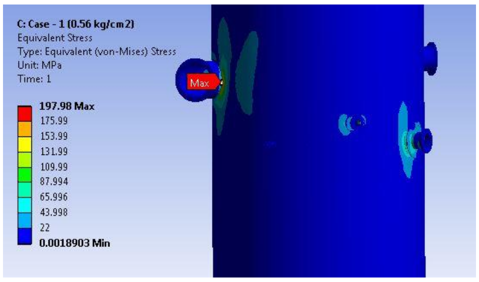

Figure 19. Maximum Stress Location (Case – I)

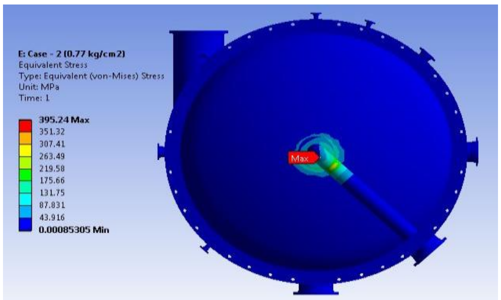

Figure 20. Maximum Stress Location (Case – II)

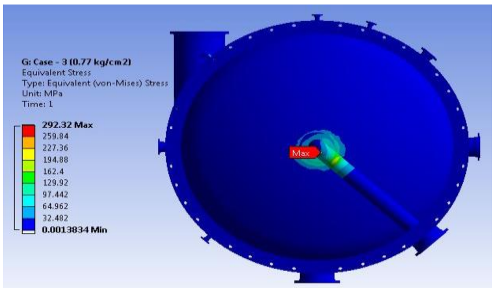

Figure 21. Maximum Stress Location (Case – III)

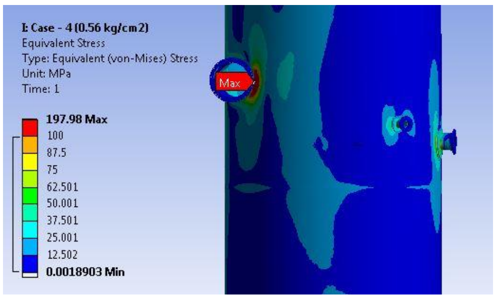

Figure 22. Maximum Stress Location (Case – IV)

The analysis results from Figures 19 - 22 show maximum stress results of the modified geometry with reference to Figure 17.

The FEA results for the remaining cases are same as case – 4 as the boundary conditions are the same.

TABLE 4. Operations in Pressure Vessel

Process/Operation	**Temperature (°C) Top/Outlet**	**Temperature (C) Bottom/ Inlet**	**Pressure (kg/cm^2g)**
Decoking Operation	40	177	0.56
Rehead and Pressure Test	40	177	0.56
Preheat	40	177	0.56

E. Fatigue Life Calculations Of Pressure Vessel:-

$$S_{alt,k} = \frac{K_f \cdot K_{e,k} \cdot \Delta S_{p,k}}{2} = S_a$$

where,

$\Delta SP, k$ = Maximum stress around the nozzle

Kf = Fatigue Strength reduction factor = 1 (ASME Section VIII, Div. 2, Table 5.11)

Ke,k = Fatigue Penalty Factor = 1.0

To calculate the design no. of cycles,

The design number of design cycles, N, can be computed from Equation (1) and Table 5 based on the stress amplitude, Sa, which is determined in accordance with part 5 of this division.

$$N = 10^x \quad (1)$$

where,

$$X = \frac{C_1+C_3Y+C_5Y^2+C_7Y^3+C_9Y^4+C_{11}Y^5}{1+C_2Y+C_4Y^2+C_6Y^3+C_8Y^4+C_{10}Y^5} \quad \ldots(2)$$

$$Y = \left(\frac{S_a}{C_{us}}\right)\left(\frac{E_{FC}}{E_T}\right) \quad ..(3)$$

The coefficient of fatigue curve for carbon steel table is taken from ASME section VIII, Division 2, Part 3

1) Fatigue Life Calculation For Original Geometry:-

From FEA maximum stress for a cycle is 427.46 MPa occurred at the nozzle and shell junction on nozzle no. 5 in case – 2 which is maximum temperature and maximum pressure case in the Pressure vessel.

TABLE 5. Coefficient of Fatigue Curve

	Coefficients of Fatigue Curve	
C_i	48 ≤ Sa < 214 (MPa) 7 ≤ Sa < 31 (ksi)	214≤ Sa < 3999 (MPa) 31 ≤ Sa < 580 (ksi)
1	2.25E+00	8.00E+00
2	-4.64E-01	5.83E-02
3	-8.31E-01	1.50E-01
4	8.63E-02	1.27E-04
5	2.02E-01	-5.26E-05
6	-6.94E-03	0
7	-2.08E-02	0
8	2.01E-04	0
9	7.14E-04	0
10	0	0
11	0	0

So the maximum stress ΔSP, k is 427.46 MPa.

TABLE 6. Calculation for S_{alt}

Sa	$S_{alt,K}$ (MPa)	213.73
Fatigue Strength reduction factor	K_f	1
Fatigue Penalty Factor	$K_{e,K}$	1
Maximum of the stress induced	$\Delta S_{P,K}$	**427.46**
Conversion factor	C_{us}	6.89
Modulus of elasticity	E_{FC}	195000
Young's modulus for material	E_T	157380

TABLE 7. Fatigue Life of Vessel

Y	38.41
x	4.16
$N = 10^X$	**14358.14**
$D_{f,k}$	0.56

2) Fatigue Life Calculation For Modified Geometry:-
From FEA of the modified geometry maximum stress for a cycle is 395.24 MPa occurred at the nozzle and shell junction on nozzle no. 5 in case – 2 which is maximum temperature and maximum pressure case in the Pressure vessel. So the maximum stress ΔSP, k is 395.24 MPa.

TABLE 8. Calculation for S_{alt}

Sa	$S_{alt,K}$ (MPa)	197.62
Fatigue Strength reduction factor	K_f	1
Fatigue Penalty Factor	$K_{e,K}$	1
Maximum of the stress induced	$\Delta S_{P,K}$	**395.24**
Conversion factor	C_{us}	6.894757
Modulus of elasticity	E_{FC}	195000.00
Young's modulus for material	E_T	157380.00

TABLE 9. Fatigue Life of Vessel

Y	35.51
x	4.22
$N = 10^X$	**16491.77**
$D_{f,k}$	0.49

4. OBSERVATION AND DISCUSSION

It is observed that the maximum stress found in case – 2 in both the geometries which is the Steam out to Blow down operation, at the nozzle and shell junction of nozzle no. 5

TABLE 10. Observation Table

	Maximum Stress for original geometry (MPa)	Maximum Stress for modified geometry (MPa)	Maximum Temp. (^oC)	Maximum Pressure (kg/cm^2 g)
Case – 1	267.64	197.98	177	0.56
Case – 2	427.46	395.24	441	0.77
Case – 3	341.35	292.32	343	0.77
Case – 4	267.64	197.98	177	0.59

TABLE 11. Fatigue Life Comparison

	Fatigue life for original geometry (Cycles)	Fatigue life for modified geometry (Cycles)
Case – 2	14358	16491

Case – 2 is the maximum temperature and maximum pressure condition case in the Pressure vessel.

It is observed that after the modification, i.e. designing the insert plate around the nozzle, the stress concentration around the nozzle is reduced which leads to increase in fatigue life of the Pressure vessel.

5. REFERENCES

1. Saidpatil, V.V. and Thakare, A.S., "Design & weight optimization of pressure vessel due to thickness using finite element analysis", ***International Journal of Emerging Engineering Research and Technology***, Vol. 2, No. 3, (2014), 1-8.
2. Brabin, T.A., Christopher, T. and Rao, B.N., "Finite element analysis of cylindrical pressure vessels having a misalignment in a circumferential joint", ***International Journal of Pressure Vessels and Piping***, Vol. 87, No. 4, (2010), 197-201.
3. Kolekar, D. and Jewargi, S., "Stress analysis of pressure vessel with different type of end connections-a review", ***International Journal of Science, Engineering and Technology Research (IJSETR)***, Vol. 3, No., (2014).
4. Niyamat, M. and K.Bicha, "Design and stress analysis of pressure vessel by using ansys", ***International Journal of Engineering Sciences & Research Technology***, Vol. 4, No. 7, (2015), 2277-9655.
5. Naser, M. and Gupta, A.V.S.S.K.S., "Structural and thermal analysis of pressure vessel by using ansys", ***International Journal of Scientific Engineering and Technology Research***, Vol. 2, No. 8, (2013), 740-744.
6. Das, S., Sundaresan, M.K. and Afia S Hameed, "Fea of cylindrical pressure vessel with different radius of opening", ***International Journal for Research in Applied Science & Engineering Technology***, Vol. 3, No. IX, (2015), 2321-9653.
7. Patil, A. and Kolhe, A., "Finite element analysis: Pressure vessel with angular leg supports", ***International Journal of Mechanical Engineering and Robotics Research***, Vol. 3, No. 3, (2014), 533-540.
8. Alashti, R.A., Rostami, S.L. and Rahimi, G., "Buckling analysis of composite lattice cylindrical shells with ribs defects", ***International Journal of Engineering-Transactions A: Basics***, Vol. 26, No. 4, (2012), 411-420.

Design and Analysis of Pressure Vessel Subjected to Pressure-temperature Variation

TECHNICAL NOTE

S. Parkhe, Annamalai K.

Mechanical and Building Sciences, VIT University, Chennai, Tamilnadu, India

P A P E R I N F O

Paper history:
Received 24 April 2017
Received in revised form 20 September 2017
Accepted 30 November 2017

Keywords:
Pressure Vessel
Finite Element Method
Fatigue Analysis
Structural Analysis
Thermal Analysis
Fatigue Life

چکیده

مخزن تحت فشار، مخزن مورد استفاده برای ذخیره سیالات تحت فشار و دمای بالاست. سیالات می توانند هوا،آب، مواد شیمیایی، سوخت، گازها و غیره باشند که بیشتر در صنایع غذایی و شیمیایی، پالایشگاه‌های نفت و غیره استفاده می‌شود. مخزن فشار در کاربردهای نیروگاهی زیر بارهای حرارتی و سازه‌ای قرار دارد. از آنجا که مخازن تحت فشار در معرض تنش‌های سازه‌ای و حرارتی قرار دارند، طراحی آنها با استفاده از روش‌های استاندارد طراحی انجام شده است. در این مقاله، روند طراحی مخزن تحت فشار در کاربرد کک سازی بر اساس استاندارد ASME، بخش VIII، کد طراحی ۱ انجام شده است. روش طراحی مطرح شده در این مقاله با روش عددی تایید شده است، به طوری که در شرایط فشار و دمای متغیر از کارافتادگی بروز نمی‌کند. مدل سازی با استفاده از SolidWorks-2015 انجام شده و تحلیل با استفاده از ANSYS صورت گرفته است.

***doi**: 10.5829/ije.2018.31.01a.09*