EVALUATION OF DESIGN PARAMETERS ON THE STRESS STATE FOR MULTI SADDLE SUPPORTED HORIZONTAL PRESSURE VESSEL

Hirenkumar Joshi[1], Ghanshyam Acharya[2]

[1] Kadi Sarva Viswavidyala, Gandhinagar -388120, Gujarat, India
[2]Atmiya Institute of Technology and Science, Rajkot-360005, Gujarat, India

Corresponding author: Hirenkumar Joshi, j_hiren@yahoo.com

Abstract: Horizontal pressure vessels supported with the saddles are subjected to high stresses near saddle vessel interface area. An accurate analysis of the local stresses and displacements of the vessels is thus of great importance. This paper analyzes the influence of design parameters in the stress distribution in both vessel and saddle for multi saddle supported pressure vessel with industrial problem. The case study of industrial autoclave has been taken and CAD model has been developed with actual dimension of saddle and vessel. The finite element analysis has been carried out in ANSYS 15.0 at hydrotest condition of autoclave and mesh sensitivity analysis has been carried out. The effects of changing the parameters such as saddle wrap angle, number of saddles, saddle width, distance from head, wear plate width, wear plate extension on the stress variations in the saddle and the pressure vessel have been analyzed.

Key words: Multi saddle supported vessels, finite element analysis

1. INTRODUCTION

The pressure vessels are having complicated design and shapes with severe changes in geometry and subjected to various loading situations. They are subjected to external and internal pressure, forces and moments due to attachments such as nozzles and also to high temperature conditions generating thermal loads. The vessels designed as per the ASME Section VIII division I code are performing well in petrochemical and mechanical industries over the years. Due to globalisation and cutting throat competition, there is severe requirement to decrease the overhead and running costs of pressure vessel and related other machinery. One of feasible way to decrease the cost is to increase the allowable stress in the ASME Code. The designers must apply sophisticated design tools with their knowledge and understanding of industrial problem to keep the factor of safety at realistic values and thus providing competent design. Finite element analysis is extensively used in the industry as it can be effectively applied for diversified and complex problems.

Fly ash brick is the product which is having tremendous potential in the construction industries due to advantages like low weight, higher compressive strength, environment friendly etc. The conventional red brick is going to phase out soon due to scarcity of raw material and government norm for environment protection.

Horizontal pressure vessels are subjected to circumferential and longitudinal stresses in addition to bending stress generated due to supports. The saddle horn area and the saddle base area are having high stress gradients. So dimension of saddle and evaluation of stress is vital in the design of pressure vessel supported by saddles. The ASME CODE [9] does not provide any guideline or rule for the design procedure for saddle. Currently over the years horizontal saddle supported vessels are designed empirically by the guideline provided by Zick [10]. Zick has applied the beam theory and made several assumptions to make simpler analysis of the problem. Zick's method does not turn out precise always because of the assumptions. The finite element method is sophisticated and robust methods for the numerical solution of complicated problems. It has been found by finite element analysis that the twin saddle supported vessels do not behave as beam for large number of cases [3]. The Zick method efficiently calculates the maximum tensile and shear stresses for twin saddles but does not give any solution when more than two saddles are required. In such case one has to use trial and error method by keeping Zick's method as reference and carryout the design procedure. In particular, the saddle produces a reaction which is highly concentrated and thereby induces localized stresses of large magnitude in the vessel, their intensity changing with vessel size and location of the saddles. An accurate analysis of the local stresses and displacements of the vessels is thus of great importance and the development of an approach used to determine these stresses has been of great interest to researchers.

Ong [7] studied the horizontal storage vessel for circumferential stress for various configurations. Ong carried out study to account for the influence of location of saddle support, support spacing, saddle wrap angle, saddle width and vessel dimensions. The study was limited to twin saddle supported vessels. Nash et al. [8] investigated the plastic collapse of horizontal saddle supported storage vessels limited to twin saddle supported vessels. Parametric study was carried out for the various factors such as saddle angle, saddle width, total length, and distance from head. Shafique M.A. Khan [2] presents the stress distribution in the horizontal vessel with application of finite element method and studied the parameters of the saddle also. His findings were strictly limited to twin saddle supported vessel only. L. Yang et al. [3] carried out parametric analysis for horizontal vessel by Finite element method and also compared the result with the experimentation done on small scaled model. N.El-Abbasi et al. [6] carried out parametric analysis of freely supported saddle. The study was based on the beam theory and finite element method. Widera et al. [5] and K. Magnucki et at., [4] also carried out analysis of horizontal pressure vessel for different range of various parameters with the help of finite element method and with a limitation to twin saddle supported pressure vessels. Xue M.D. et al. [14] effectively used ANSYS for carrying out FEA and found good agreement between the results obtained by thin shell theory and FEA. Xue L. et al. [16] validated the results of FEA with experimentation with good agreement and further developed parametric equations covering wide range of geometic parameters for the assessment of burst pressure. Fang J. et al. [15] performed the comparative study of strength behaviour of vessels with pad reinforcement in nozzle surrounding with different diameter ratios by finite element method and also carried out testing on fabricated models. Ng H.W. et al. [11] investigated the fixed saddle for storing high temperature liquid for thermal stresses by applying finite element method and also futher validated the predictions experimentally.

With a consideration of above finding by various researchers it can be seen that all the studies are related more towards the stresses generated in the pressure vessel and the saddle supports have not been given good attention. Further all the work has been done for twin saddle supported vessels only.

This study aims to study the influence of design parameters in the stress distribution in both vessel and saddle for multi saddle supported pressure vessel with industrial problem. The case study of industrial autoclave has been taken and CAD model has been developed with actual geometry of autoclave. The model has been simulated with actual boundary conditions of hydrotest condition and finite element method has been applied in ANSYS 15.0 to analyze the effect of various parameters on the stress distributions both in the saddle and the pressure vessel.

2. PROBLEM SETUP

2.1 Autoclave Design Data

Fly ash brick needs 21 days curing time if done in open atmosphere. Autoclaves are used cure fly ash bricks at the faster rate. The total curing time is reduced to 6-8 hours from 21 days by curing the bricks under pressure and temperature provided by steam into the autoclaves. Autocalves are large horizontal cylindrical pressure vessel supports on multiple supports due to its long length. The case study has been taken with MetalFab Engineers Surat. The autoclave has been designed as per code ASME SECTION VIII DIV. 1 and following dimensions are obtained, figure 1, tables 1 and 2.

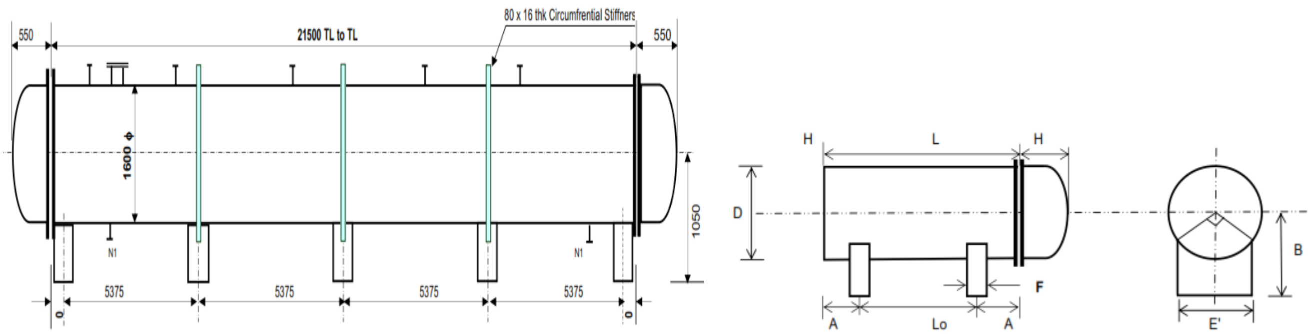

Fig. 1 Fly ash brick autoclave and side view of fly ash brick autoclave

Table 1. Design data

Sr.No.	Vessel Design Data	
1	Inside diameter	1600 mm
2	Overall length	21500 mm
3	Design pressure	13 kg/cm^2
4	Hydrotest pressure	16.9 kg/cm^2
5	Design temperature	200° c
6	Material of vessel	SA516 Gr 70
7	Material of saddles	IS 2062 GR.02
8	Shell and head thickness	12 mm

Table 2. Geometric data

Sr.No.	Vessel and Saddle Geometric Data	
1	Centre height of vessel B	1050 mm
2	Wrap angle of Saddle	120°
3	Saddle breadth E'	1400 mm
4	Wear plate thickness	10 mm
5	Saddle width	200 mm
6	Web and rib thickness	10 mm
7	Number of ribs required	4
8	Wear plate width	300 mm

2.2. Finite element analysis

CAD Model of vessel and saddle has been developed in Creo 2.0 Parametric and has been exported to ANSYS 15.0. Ansys 15.0 has been used for carrying Finite Element Analysis of the pressure Vessel and Saddle. For carrying out FEA 8 node brick element has been used. The boundary condition with respect to hydrotest condition has been applied for simulation as it is the most stressed condition for the vessel. All the saddles have been considered as fixed support. The self weight of the pressure vessel has also been taken into account.

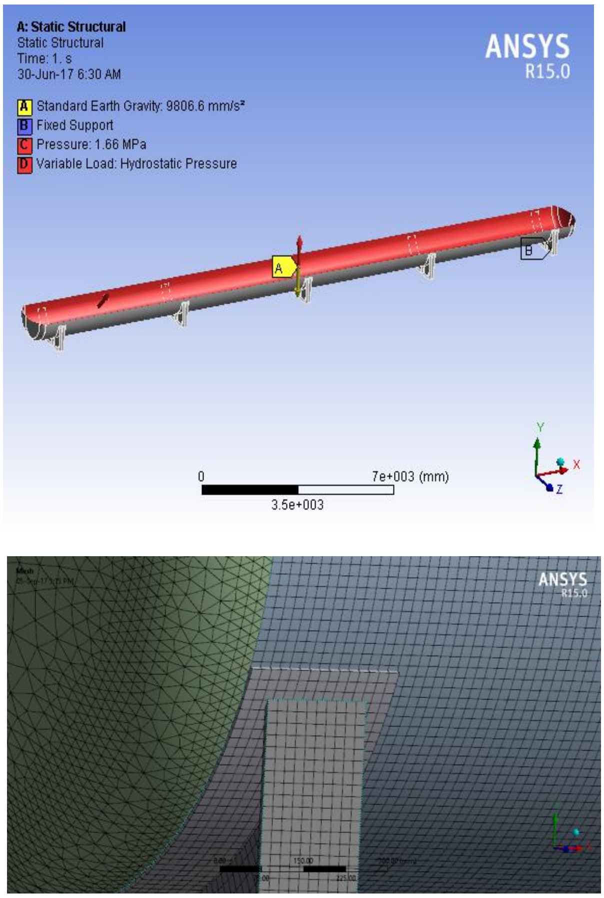

Fig. 2. Boundary condition and meshing near saddle horn area

2.3. Mesh sensitivity analysis

In finite element analysis to check the reliability of the result mesh sensitivity analysis has been carried out. Meshing has been carried out by mapped meshing by giving the number of element along the various edges of vessel and saddle. The mesh has been refined by gradually increasing the number of element and convergence has been reached as shown in the table 3.

Table 3. Convergence

Sr. No	Solution name	No. of element	Equi. Stress at first saddle (MPa)	% diff.
1	basic	716363	139.08	-
2	Basic_1	1425793	186.92	34.39
3	Basic_2	2450603	204.36	9.33
4	Basic_3	3381149	220.5	7.89
5	Basic_4	4170206	225.63	2.33

The convergence has been achieved when the percentage difference in the stress values are less than 3 percent for the consequent mesh refinement. All the simulations have been carried out as per the mesh setting received at the convergent solution.

2.4. Parameters

Various parameters affecting the stress distribution in the pressure vessel and saddle are as listed below in table 4.

Table 4. Factors affecting stress distribution

1	Saddle wrap angle
2	WEAR Plate Extension
3	Distance of Saddle from Head
4	Wear Plate Width
5	Saddle Width
6	Shell thickness
7	Internal Pressure
8	Hydrostatic Pressure
9	Wind Force
10	Self Weight
11	Sesimic Force
12	Vessel Diameter
13	Vessel Length
14	Wear Plate Thickness
15	Saddle Height
16	Rib Thickness
17	Number of Rib
18	Thickness of Base Plate

The parameters shown in table 4 are checked for their effect on the stress initially. The parameters as shown in the table 5 have been finally shortlisted to check for the influence. Their range has been selected carefully from the various references Tooth [1], Abbasi [6], Ong [7], Duthie [12], Ong [13] and Nash [8] and in consulation with experts from industries. The simulations have been carried out by changing the one parameter at time and results have been obtained. The graphs have been generated for the stress values at 1^{st} and middle saddle at the saddle interface area in the vessel and at the base in the saddle.

3. EFFECT OF PARAMETERS ON STRESS DISTRIBUTION

The effect of parameters, table 4, has been investigated for 1st saddle and mid saddle and at the saddle horn area and saddle base area. Saddle horn area is the area in the vessel where wear plate just adjoins the vessel. In this area high stress are produced compared to the other vessel area. The second area is the saddle base in the rib of saddle supports.

Table 5. Parameters and its range for simulation in FEA.

Sr. No	Parameters	Range of Parameters
1	Number of saddles	3, 4, 5
2	Wrap angle(ɵ)	120, 140, 160, 180
3	Saddle width (mm)	200, 250, 300
4	Wear plate width (mm)	300, 400, 500, 600
5	Distance from head (mm)	1100, 1500, 1900, 2250
6	Wear plate extension (ɵ)	0, 6, 14

3.1. Effect of number of saddle

The effect of reducing the number of saddle is to increase in value of stress near horn area both in first and mid saddle but the generated stresses do not cross the yield point so one can definitely reduce the number of saddle (figure 3). In the saddle at the base the stress generated increase with the reduction in number of saddle but no effect in the mid saddle is observed. Thus this parameter can be reduced and the reducing number of saddle will affect greatly on cost since entire saddle is reduced.

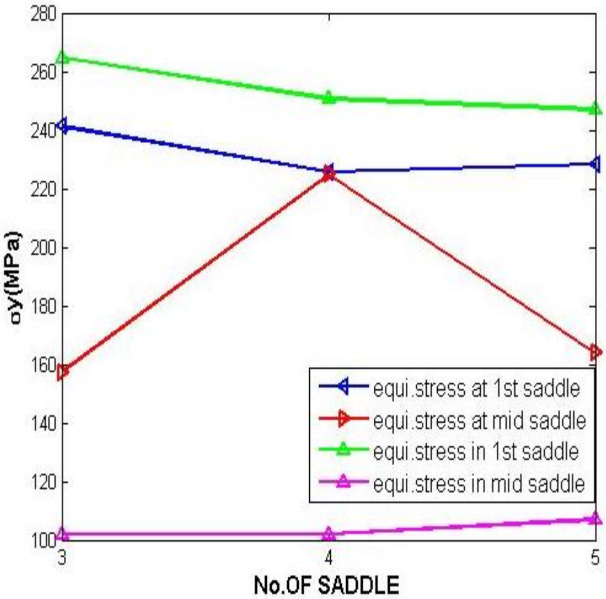

Fig. 3. Effect of number of saddle on maximum stress

3.2. Effect of wrap angle

The results shows a considerable reduction in stress for 140° wrap angle at the mid saddle but again rise in stress value at the first saddle. Maximum reduction of stress in the vessel is possible by using wrap angle near to 180° which generates more stress in the saddle base (figure 4). So increasing the saddle wrap angel transfer the stress from the saddle horn area to saddle base area in the rib. Wrap angle around 160° generates overall less stress in both vessel and saddle. So saddle wrap angle should be carefully selected for uniform stress distribution.

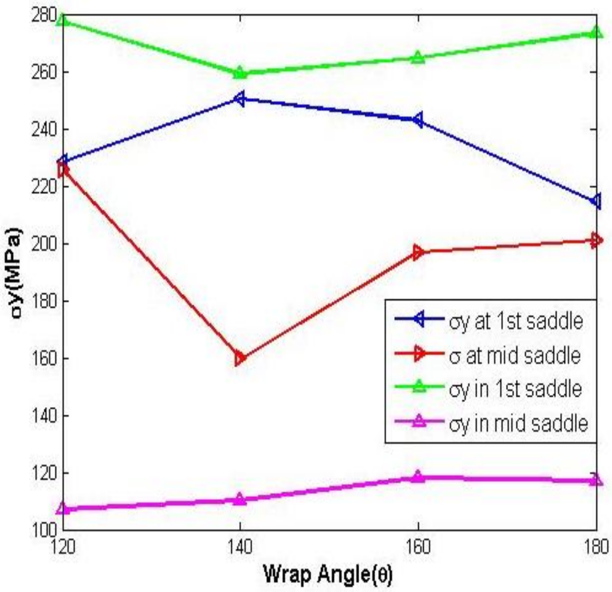

Fig. 4. Effect of wrap angle on maximum stress

3.3. Effect of saddle width

In the saddle horn area of first saddle maximum stress increase as width of saddle increases while in the mid saddle maximum stress decrease as width increases (figure 5). In the saddle base area maximum stress decreases at width increase for first saddle and vice versa for mid saddle.

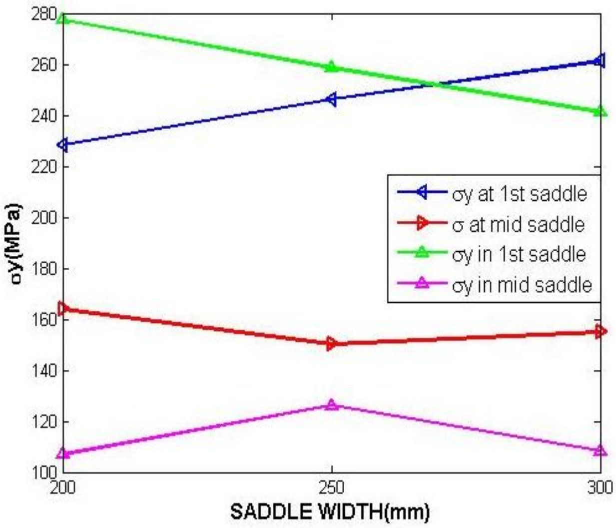

Fig. 5. Effect of saddle width on maximum stress

3.4. Effect of wear plate width

In the saddle horn area max. stress is considerably reduced with increase in the width of wear plate for both first and mid saddle with a minimum value at the width near to 500 mm (figure 6). Further max. stress in the saddle base region of first saddle is reduced considerably for saddle width of 500 mm.

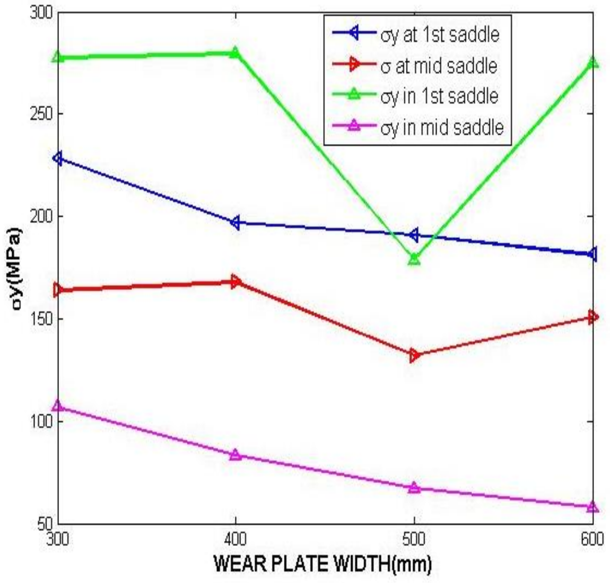

Fig. 6. Effect of wear plate width on maximum stress

3.5. Effect of distance from head

Different A/L ratios from 0.025 to 0.1 have been investigated and max. value of stress in horn area of first saddle is observed for the ratio 0.05 . Further increase in the ration results in increase in stress in (figure 7) the saddle horn area. Further reduction in stress in the horn area at the ratio 0.05 results an increase in stress in the saddle base region.

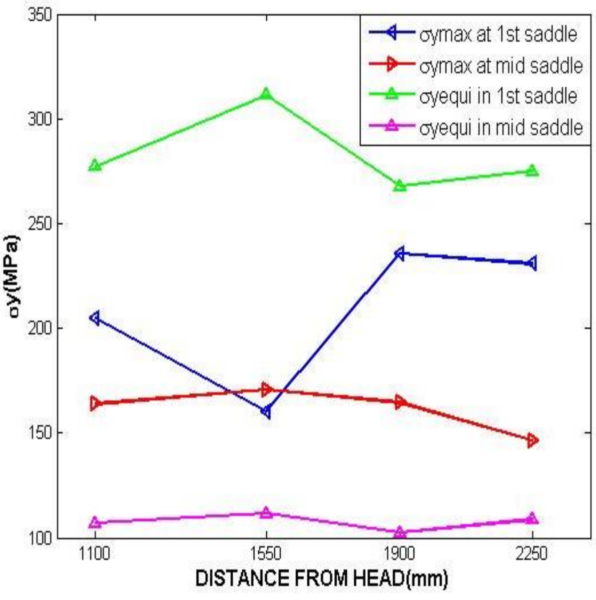

Fig. 7. Effect of distance from head on maximum stress

3.6. Effect of wear plate extension

Wear plate extension between 6° to 14° gives very enthusiastic results for reduction in max. Stress in saddle horn area (figure 8). Further this reduction is not related with increase in stress in saddle base area like in parameters distance from head and wraps angle. So the parameter wear plate extension can be very effectively for reduction of stress in vessel and saddle.

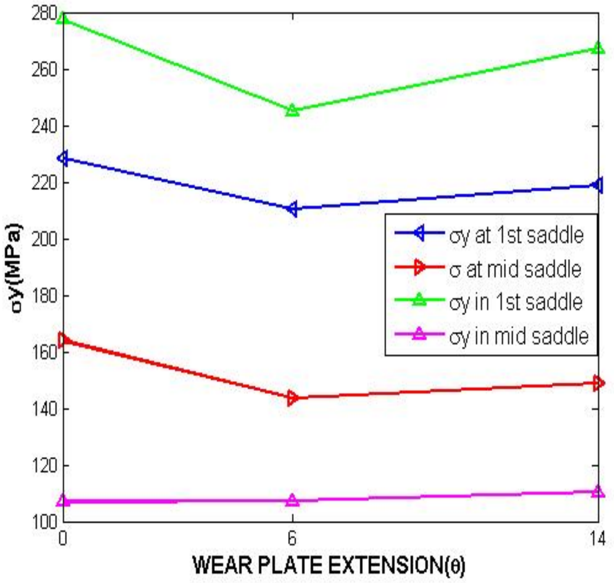

Fig. 8. Effect of wear plate extension on maximum stress

4. CONCLUSIONS

The effects of changing various design parameters on von Mises stresses have been evaluated numerically at saddle horn area and at the rib of saddle of pressure vessel. The peak values of stresses are in the saddle horn ranging from 160 MPa to 230 MPa and in the rib of the saddle ranging from 180 MPa to 310 MPa for various design parameters. Effect of changing wear plate extension and wear plate width reduce the stress nearly 15 percentage while the effect of changing wrap angle results in reduction about 20 percentage. The reduction in stress is due to the more support region provided in the saddle. The saddle wrap angle, saddle width and distance of head factors plays moderate role in reducing maximum stress. The number of saddle can be reduced as vessel and saddle does not exhibit any violence in yield limit and further generated stresses can be reduced by combination of various saddle design parameters.

5. REFERENCES

1. Tooth A.S., Chan, G.C.M. and Spence J, and Nash D H,(2003) *Horizontal Saddle Supported Storage Vessels: A Parametric Study of Plastic Loads*. Pressure Equipment Technology: Theory and Practice, Professional Engineering Publishing Ltd., London and

Bury St. Edmonds, UK, ISBN 978-1-86058-401-5,109-138.
2. Shafique M. A. Khan (2010), *Stress distributions in a horizontal pressure vessel and the saddle supports*, International Journal of pressure vessel and piping 87, 239-244.
3. Yang L., Weinberger C., Shah Y.T., (1994), *Finite Element Analysis on horizontal vessels with saddle supports,* Journal of computers & structures 52(3), 387-395.
4. Magnucki K., Stasiewicz P., Szyc W., (2003), *Flexible saddle support of a horizontal cylindrical Pressure vessel*, International Journal of Pressure Vessel and Piping, 80, 205-210.
5. Widera G.E.O., Sang Z.F., Natarajan R., (1988), *On the Design of Horizontal Pressure Vessels,* Journal of Pressure Vessel Technology,110,393-401
6. El-bbasi N., Meguid S.A., Czekanski A., (2001), *Three dimensional finite element analysis of saddle supported pressure vessels,* International Journal of Mechanical Sciences, 43, 1229-1242.
7. Ong Lin Seng (1991), *Parametric Study of peak circumferential stress at the saddle* support, International Journal of Pressure Vessel and Piping, 48, 183-207.
8. Nash D. H., Spence J., Tooth A.S. (2003), *On the plastic collapse of horizontal saddle supported storage vessels,* Proceddings of the 10th International Conference of Pressure Vesssel Technology, Design by Analysis, 73-78, Vienna, Austria.
9. ASME'S Boiler and Pressure Vessel Code (BPVC), (2013), Section VIII, Division 1, 6-9.
10. Zick L.P., (1951), *Stresses in large horizontal cylindrical pressure vessels on two saddle support.* Weld J Res Suppl, 30, 435-445.
11. Ng H.W., (1999), H*orizontal vessels on fixed supports used for storing hign temperature liquid: An experimental validation of finite lement modeling*. Journal of strain analysis for engineering design, 34, 83-96.
12. Duthie G., White G.C., Tooth A.S., (1982), *An analysis for cylindrical vessels under local loading-application to saddle supported vessel problems*, Jour. of Strain Analysis,17(3), 157-166.
13. Ong L.S., Tooth A.S., Cheung J.S.T., Ng H.W., (1998), *Parametric equations for maximum stressess in cylindrical vessels subjected to thermal expansion loading*, International Journal of Pressure Vessel and Piping, 75, 255-262.
14. Xue M.D., Li D.F., Hwang K.C., (2004) *Analytical solution of two intersecting cylindrical shells subjected to transverse moment on nozzle* International Journal of solids and structures, 41, 6949-6962.
15. Fang J., Tang Q.H., Sang Z.F., (2009), *A comparative study of usefulness for pad reinforcement in cylindrical vessels under external load on nozzle*, International Journal of Pressure Vessels and Piping, 86, 273-279.
16. Xue L., Widera G.E.O., Sang Z., (2010), *parametric FEA study of Burst Pressure of Cylindrical Shell*, Journal of pressure Vessel Technology, 132, 031203-031210.

Received: July 6, 2017 / Accepted: December 10, 2017 / Paper available online: December 20, 2017