A Review Paper on Pressure Vessel Design and Analysis

Shyam R. Gupta,
PG Student of Mechanical Department
Kalol Institution of Technology And Research Center, Kalol, India.

Chetan P. Vora
Associate Professor, Mechanical Department Kalol Institution of Technology and Research Center, Kalol, India.

***Abstract*— This paper reviews some of the developments in the determination of stress concentration factor in pressure vessels at openings, stress analysis of different types of end connections and minimization stress with the help of optimize location and angle of nozzle on shell and head. The literature has indicated a growing interest in the field of stress concentration analysis in the pressure vessels. The motivation for this research is to analyze the stress concentration occurring at the openings of the pressure vessels and the means to reduce the effect of the same. Design of pressure vessels is governed by the ASME pressure vessel code. The code gives for thickness and stress of basic components, it is up to the designer to select appropriate analytical as procedure for determining stress due to other loadings. In this paper the recent and past developments, theories for estimation of stress Concentrations are presented and the scope for future studies is also presented.**

Keywords*— *Pressure Vessel, ANSYS, PVElite Software, PRO-E, Mechanical Stresses, ABAQUS

I. INTRODUCTION

Pressure vessels find wide applications in thermal and nuclear power plants, process and chemical industries, in space and ocean depths, and fluid supply systems in industries. The failure of pressure vessel may result in loss of life, health hazards and damage of property. Due to practical requirements, pressure vessels are often equipped with openings of various shapes, sizes and positions. Vessels have openings to accommodate manholes, handholds, and nozzles. Openings vary in size from small drain nozzles to full vessel size openings with body flanges. The openings cannot be avoided because of various piping or measuring gauge attachments. They allow for the mounting of equipment, the insertion of instrumentation, and the connection of piping facilitating the introduction and extraction of content but they also lead to the high stress concentration which leads to the failure of pressure vessel. Openings in pressure vessels are frequent, in fact all riveted constructions make use of such means of fabrication, and all vessels must have openings. These geometric discontinuities alter the stress distribution in the neighborhood of discontinuity so that elementary stress equations no longer prevail. Such discontinuities are called stress raisers and the regions in which they occur are called the areas of stress concentrations.

II. LITERATURE REVIEW:

In this section research papers are discussed related to the present work. Published papers are highlight in this section.

A. *M Javed Hyder and M Asif*[1]

etc presented work to optimize location and size of opening in pressure vessel cylinder using ANSYS, Analysis performed for three thick-walled cylinders with internal diameter 20, 25 and 30 cm having 30 cm height and wall thickness of 20 mm. first they done analysis of pressure vessel cylinder without hole, they found tangential, longitudinal, radial, and von misses stress ,then optimization of hole size is carried out by making hole having diameter of 4,8,10,12,14,16 and 20 mm located at center in each of three thick cylinders, from fig 14 they found that the optimum size of hole is 8 mm cylinder having internal diameter 20 cm and hole size 10 mm for cylinder having internal diameter of 25 cm and 30 cm has lowest von Misses stress value and finally 12 mm hole located at 1/6, 1/8, 2/8, 3/8, and 4/8 of cylinder from top in all three cylinders, and they found Von Misses stress is maximum at the center 0.500 location and decreased directed away from the center and the stress increased at the location change from 0.1250 to 0.0625 from cylinder top due to end effect and finally they found Von-Mises stress is minimum at location 1/8 of cylinder height.

B. *Josip Kacmarcik, Nedelijko Vukojevic And Fuad Hadzikadunic*[2]

etc concluded that comparison show good agreement between the stress concentration factor determined with two different method, here two different method strain gauge with experimental set-up and finite element analysis with ABAQUS software are used for two different nozzle geometries investigation, here two stress concentration factor defined by maximum principal stress and maximum von-Mises stress are calculated by strain gauge measurement and compared with ABAQUS software, in this paper nozzle external radius are different C_1 Nozzle has higher radius then C_2 nozzle, but both nozzle have same thickness of vessel wall and external radius of a vessel, in this paper only 1/8 of the vessel part and ¼ of nozzle part is modeled because it is possible to defined three symmetry planes and here as a mesh generation 3D tetragonal elements are implemented, stress concentration factor is obtained by the value of stress(principal and von Mises) obtained via FEM analysis and strain gauge measurement, when compared both method it is shown that the maximum deviation of 15.5% is acceptable for engineering application of stress concentration factor and FEM analysis is very reliable enough for determining stress concentration factor in pressure vessel design. And this research also show advantages of FEM analysis in possibility to determine stresses on vessel internal side that can be greater than external stresses which is very difficult for strain gauge measurement.

C. V.N. Skopinsky, A.B. SMETANKIN[3]

presented work on modeling and stress analysis of nozzle connection in Ellipsoidal head of pressure vessel under external loading, in this paper he used Timoshenko shell theory and the finite element method, the effect of stress concentration in external loading has more effect than in the internal pressure, there is an appreciable increase of the maximum stress for shell in the interaction region even at the small level of nominal stress, non-radial and offset connection have non-uniform distribution of stress on the interaction curve between the nozzle and the head, the influence of angular parameter α for non-radial nozzle connection is shown in this paper, a decrease of maximum effective stress as an angle α increase is more significant for non-central connection, and in case of torsional moment loading, the angle affects the stress in opposite manner, the stress in the shell increase as alpha angle increase.

D. J. Fang,Q.H.Tang[4]

etc presented work on a comparative study of strength behavior for cylindrical shell interaction with and without pad reinforcement under out-of-plane moment loading on nozzle, three pairs of full-scale test vessel with different mean diameter of nozzle to mean diameter of cylindrical vessel ratio were designed and fabricated for testing and analysis ,the material of the cylinder, reinforcement pad and the nozzle are low carbon steel, result from this research indicate that the maximum elastic stress and stress ratio are reduced by pad reinforcement, they found that in test reduction rate is 20-60% and in finite element analysis reduction rate is 28-59% and its rate of reduction depend upon structure and dimension of the vessel for example D/d ratio, and result also indicate that the plastic limit of nozzle in cylinder vessel is increased by pad reinforcement, generally rate of increase is about 40-70% from test and its larger than 40% from finite element analysis, so the conclusion given from the result that the reinforcement structure are useful under static external load on nozzle.

E. Pravin Naral and P S Kachare[5]

presented work on structural analysis of nozzle attachment on pressure vessel design, they said if the nozzle is kept on peak of the dished end it do not disturb the symmetry of the vessel, but if it is placed on the placed on the periphery of the vessel, it may be disturb the symmetry of the vessel. Size, diameter, angle, etc of nozzle connection may significantly vary even in one pressure vessel, these nozzle cause geometric discontinuity of vessel wall, so a stress concentration created around a opening, the junction may fail due to high stress ,so detailed analysis is must be required, in this paper conduct a study analysis, what will be the effect of the nozzle angle and increase number of nozzle on the periphery of pressure vessel until the symmetry is achieved, and find out optimum angle such that the stress are maintained within limits. in this paper first one nozzle placed on top on shell and calculated stresses with finite element analysis, then two nozzle placed with angle 60 degree from each other, then again two nozzle placed at angle 90 degree from each other, then also again two nozzle placed at angle 180 degree from each other ,then three nozzle placed at angle 60 degree from each other, then again three nozzle placed at angle 90 degree from each other, then four nozzle placed at angle 60 degree and again four nozzle placed at 90 degree from each other and calculated stress from ANSYS software ,from this study they found the result that peak stresses for symmetrical nozzle attachment is lowest than the others and stress increment factor for symmetric nozzle attachment is lower than other, here the stress value is minimum at two nozzle which is placed at angle 180 degree and four nozzle placed at angle 90 degree from each other, this state that the symmetry nozzle attachment had always lower stress than others.

F. James j Xu, benedict C. Sun, Bernard Koplik[6]

had did work on local pressure stress on lateral pipe-nozzle with various angle of interaction, this paper report variation of local pressure stress factor at the junction of pipe-nozzle when its angle varies from 90 to 30 degree, the circumferential and longitudinal stress at four symmetric points around the pipe-nozzle junction are plotted as function of an angle, the ALGOR finite element software was employed to model for the true pipe-nozzle geometry, the numerical stress result come from parameters beta and gamma which are the nozzle mean radius and pipe thickness, at angle 90 degree at this angle result had low value local stress, these stress increase as angle of interaction is decrease from 90 degree and stress value more decrease when angle is decrease from 45 degree, the inside crotch point B has worst circumferential stress value, and concluded that angle 90 degree local pressure stress are same at point A and B as same as point C and D due to symmetry. And it had low stress value than other angle

G. Amran Ayob[7]

worked on stress analysis of torispherical shell with radial nozzle, in this paper experimental reading was taken with help of 0.0625-inch foil string gauge which was bonded to the outer and inner surface of the shell, the model was instrumented with 39 pairs of 0.0625-inch foil strain gauge, these gauge was located between S=-0.1 to S=0.5 in the meridional direction. The experimental result used here is the part of test programmer carried out by drabbles to determine the shakedown behavior of a torispherical vessel with nozzle, under action of internal pressure, thrust and bending moment applied to the nozzle. There are three interacting geometric location which could influence stress field, the maximum stress could occur any of sphere-nozzle, sphere-knuckle and cylinder-knuckle junction the graph of the elastic stress factor distribution along meridional plane due to four load case shown in this paper, the crotch corner and the weld-crown region are the highest stress area with ESF approximately 2, *H. V. N. Skopinsky*[8]

had worked on stresses in ellipsoidal pressure vessel heads with noncentral nozzle, the objective of this paper is more investigation of shell intersection problem, the shell theory and finite element method are used for stress analysis of nozzle connections in ellipsoidal heads of the pressure vessel, here nozzle is considerably displaced on ellipsoidal head from head axis is considered in this paper, the feature of numerical procedure, structural modeling of nozzle-head shell intersections and SAIS special-purpose computer program are discussed. The result of stress analysis and parametric study of ellipsoidal vessel head with a noncentral nozzle under internal pressure loading are presented, in many practical design, the nozzle is placed at a relatively large distance from the head axis. Special consideration of these case is given in this analysis, this stress analysis result better understanding of this poorly investigated problem and give the possibility of achieving a more reliable design of nozzle connections on the pressure vessel heads, also the SAIS program can be used for design optimization purpose e.g. nozzle location finding.

I. Jaroslav Mackerle[9]

had worked on bibliographical review of finite element method(FEMs) applied for the analysis of pressure vessel structural/components and piping from the theoretical as well as practical points of view, he searched paper contains 856 reference to papers and conference proceeding on the subject that were published in 2001-2004, he found papers those are classified in the following categories: linear and nonlinear, static and dynamic, stress and deflection analysis, stability problem, thermal problem, fracture mechanics problem, contact problem, fluid-structure interaction problem: manufacturing pipe and tube: welded pipe and pressure vessel components: development of special finite element for pressure vessel and pipes, finite element software and other topics, and he found that linear and nonlinear, static and dynamic, static and deflection analysis and fracture mechanics problem had various topic in pressure vessel and piping.

J. P balicevic, D Kozak, D. Karlievic[10]

presented work on ANLYTICAL and NUMERICAL solution of internal forces by cylindrical pressure vessel with semi-elliptical heads, in this paper the solution for internal forces and displacement in the thin-walled cylindrical pressure vessel with ellipsoidal head using general theory of thin walled shell of resolution have been proposed, distribution of the forces and displacement in thin walled shell are given in mathematical form, finite element analysis of the cylindrical vessel with semi-elliptical head has been done by using ANSYS 10 code for to confirms analytical solution, here ellipsoidal head model made as axi-symmetric problem to avoid bending effect on the contact between heads and cylinders and author concluded principal stresses calculated analytically are very close to the finite element result(the difference is less than 3%)

K. M F hsieh, D G Moffat, J mistry[11]

had worked on nozzle in the knuckle region of a torispherical head, in this paper limit load interaction plot for pressure Vs nozzle axial force, in-plane moment, out-of-plane moment and for in-plane moment versus out-of-plane moment are also present, here six model included with nozzle offset location nozzle offset/vessel outer diameter in present study, model 1 is the ax symmetric case with nozzle located in the center of the crown, the model 3 offset the outermost weld location is at crown/knuckle junction and in this work FE model was created with using PATRAN mesh generation program and stress analysis work was done by using ABAQUS program, they concluded that the nozzle has very little influence on the limit pressure of the head, even when it is located in the knuckle region of the head, for external load applied to the nozzle, the effect of increasing the offset is to increase the limit loads.

L. B.S.Thakkar and S.A.Thakkar [12]

did a case study and put efforts to design the pressure vessel using ASME codes & standards to legalize the design. The performance of a pressure vessel under pressure can be determined by conducting a series of tests to the relevant ASME standard in future scope they have mentioned Design of pressure vessel in PVELITE software can be accrue. Further FEA analysis can be done to verify the above design procedure, they concluded that the design of pressure vessel is more of a selection procedure, selection of its components to be more precise rather designing of every components, pressure vessel components are selected on the basis of available ASME standard and the manufactures also follow the ASME standard while manufacturing the components so that leaves designer free from designing the components. This aspect of design greatly reduce the development time of new pressure vessel, it also allows the designer to keep free from multiple prototype for pressure vessel before finalizing the design, here standard part are used so it reduce time for replacement so less overall cost

M. Shaik Abdul Lathuef and K.Chandra Sekhar [13]

discusses some of the potential unintended consequences related to Governing Thickness of shell as per ASME. Here have a scope to change the code values by take the minimum governing thickness of pressure vessel shell to the desired requirements and also relocate of nozzle location to minimize the stresses in the shell. In this paper nozzle located at five places and analysis with ANSYS here nozzle locates at shell left end, at the shell middle, at the shell right end, at dished end of both side and calculate stress. And they found from result that the stress would be Minimum at the dished end with hillside orientation. A low value of the factor of safety results in economy of material this will lead to thinner and more flexible and economical vessels. Here we evaluated the stress in the vessel by Zick analysis approach.

N. Binesh P Vyas, R. M. Tayade[14]

concluded that Design of pressure vessel by using PVElite gives accurate analysis result and also reduces time .A vertical pressure vessel has been designed using graphical based software named PVElite. For designing of vertical leg supported pressure vessel some input parameters like volume, inside diameter, design pressure (either inside pressure or external pressure), temperature, material, processing fluid. Etc. is required. PVElite gives thickness of shell, thickness of head, height of head, thickness of nozzle, manhole, PVElite calculate local stress according to welding research council

(WRC) 107, further research need to explore environmental parameter such as earthquake, thermal load, fluctuation load and so on.

O. Dražan Kozak Ivan Samardžić [15]

etc had worked on stress analysis of cylindrical vessel with changeable head geometry, The main objective of this paper is numerical analyses of cylindrical pressure vessel with changeable head geometry (semi-elliptical and hemispherical heads) and comparison of results in means of precision and time needed for getting the solution, comparison of analytical and numerical results for pressure vessel with hemispherical heads is shown, In this paper a numerical analysis of pressure vessel with hemispherical and semi-elliptical heads is performed, with three types of elements: SOLID 95, PLANE 183 and SHELL 181. It is concluded that in both cases of pressure vessel heads, using of PLANE 183 element presents the best approach, because of minimal number of elements for meshing, shortest calculation time, this type of axsymmetric element could be recommended in such cases, when the total symmetry of model is considered.

P. Yogesh Borse Avadesh Sharma[16]

presented work on Modeling of Pressure Vessels with different End Connections using Pro Mechanica, author used here only three types head hemispherical, elliptical and torispherical In this paper authors, describes its basic structure and the engineering finite element modeling for analyzing, testing and validation of pressure vessels under high stress zones, Analysis results do not suggests the use of Toro spherical end connection with the same thickness due to more displacement occurs in this type of head.

Q. Vikram V. Mane, Vinayak H .Khatawate [17]

etc done their work on stress analysis of Ellipsoidal head pressure vessel with the help of finite element analysis and experimental work, they used electrical strain gauges for strain measurement and compared result with ANSYS software .and they found the results of the stress analysis by classical methods are more than the actual stresses measured by strain gauges and less than the finite element analysis.

R. Ugˇur Gu¨ven [18]

obtained the failure pressures of thick and thin walled cylindrical pressure vessels considering the Voce hardening law and plastic orthotropic effect. The solution presented is used to compare the failure pressures of copper and brass cylindrical pressure vessels. The failure pressures of thick and thin walled cylindrical vessels are solved by numerical and closed form solutions. The solutions presented are used to compare the failure pressures copper and brass cylindrical vessels.

S. Pavo Baličević[19]

proposed method for stress analysis in cylindrical pressure vessels with ellipsoidal heads, based on the ax symmetric shell theory. The starting point were the approximate solutions of the differential equation system that were used to get mathematical expressions for determining internal forces, moments and displacements in the vessel walls. Application of the method was shown on a selected numerical example, while a special computer programmed was created for calculation purposes.

T. Shafique M.A. Khan [20]

done analysis and shown result of stress distributions in a horizontal pressure vessel and the saddle supports. The results are obtained from a 3D finite element analysis. A quarter of the pressure vessel is modeled with realistic details of saddle supports. Physical reasons for favoring of a particular value of ratio of the distance of support from the end of the vessel to the length of the vessel are also outlined.

U. Bandarupalli Praneeth, T.B.S.Rao[21]

compared The theoretical values and ANSYS value for both solid wall and multilayer pressure vessels. And they concluded that multi layered pressure vessels are superior for high pressures and high temperature operating conditions over the conventional mono block pressure vessel. Theoretical calculated values by using Different formulas are very close to that of the values obtained from ANSYS analysis is suitable for multilayer pressure vessels.

V. Drazan kozak[22]

etc presented worked on Overloading Effect on the Carrying Capacity of Cylindrical Tank with Torispherical Heads for the Underground Storage of Petrol, Horizontal cylindrical double skin steel tank with torispherical heads for the underground storage of petrol has been manufactured, Before exploitation it has to be tested with pressure of 2 bars according EN 12285-1 norm. During the pressurization uncontrolled pressure increasing happens. Effects on this overloading have been analyzed by using of finite element method.

W. M. Pradeep Kumar, K. Vanisree [23]

etc presented work on Design and Implementation of Circular Cross Sectional Pressure Vessel Using Pro-E and ANSYS, EW system frame assembly is a leak proof and contains high pressure to hold precious electronic at a pressure different from the ambient pressure, they concluded the stresses developed in the circular cross section with hemispherical end caps are very less as compared to rectangle cross sectional vessel which is used in submarine EW system. Also in the circular cross section, the stresses and deflections are minimum. Design of pressure vessel done by ASME code section VII and analysis is done by ANSYS software.

X. L. P. Zick [24]

indicates the approximate stresses that exist in cylindrical vessels supported on two saddles at various locations. Knowing these stresses, it is possible to determine which vessels may be designed for internal pressure alone, and to design structurally adequate and economical stiffening for the vessels which require it. Formulas are developed to cover various conditions, and a chart is given which covers support designs for pressure vessels made of mild steel for storage of liquid weighing 42 lb. per cu. ft.

Y. Nishant M. Tandel, Jignesh M parmar [25]

had presented work on A review on pressure vessel design and analysis, this paper deal with vessel are subjected to various applied forces acting in combination with internal or external pressure and some design principle, design of pressure vessel is governed by ASME pressure vessel code, design of different pressure vessel concerned with element such as shell, dish end, operating man hole, support leg, based on standard and code and evaluation of shell and dish end analyzed by means of analysis, and this paper they concluded that finite element analysis is an extremely powerful tool foe pressure vessel and also concluded the design method to be used in pressure vessel are depend upon stresses and internal or external pressure

Z. Modi A J, Jadav C.S[26]

concluded that the radial stresses in case of hemispherical head pressure vessel is low compared to other types of head, in this paper author study the comparative structural behavior of different types of geometry of pressure vessel, the head is under internal uniform pressure, the analytical and finite element method used for finding stresses in pressure vessel, the aim is finding best head for specific parameter with finite element analysis of thin cylindrical pressure vessel, here three types of geometry consider like hemisphere, flat and ellipsoidal and computation result compared with finite element analysis.

AA. Hardik B nayak, R R trivedi[27]

had worked on stress analysis of reactor nozzle to head junction, in this paper stress analysis is carried out for nozzle to head junction subjected to applied external load, internal pressure and moments under different loading condition, the stress will occur at the nozzle to head or shell junction area due to discontinuity of geometry defect will occur and the junction region will be the failure source of the whole structure, stresses at reactor nozzle to head junction are obtained by welding research council(WRC) 107 and PVcode cal software 2008 with and without stress indices and they found that general solution obtained for nozzle to head junction have not given the result in allowable limits for WRC 107 and PVcode cal software because it does not take pressure into account while calculating the local and general primary membrane equivalent stress and the stress calculated by nozzle to head junction using ANSYS software give more accurate data than 107.

BB. M J Mungla [28]

had conducted design and analysis of various components of pressure vessels like shell, heads, flanges, and nozzle and support structures along using ASME code. Design of base ring and skirt sections has not been covered under ASME code and their dimensions are calculated with general design principles. Stress analysis of these components has been carried out with combined load cases,

III. CONCLUSIONS:

From the literature review it is seen that ASME and other code are providing solutions for more general cases and required higher factor of safety, also limit load and stress concentration formulae are not available for non standard shape and intersection and geometrical discontinuity, most of researcher have worked in thin-pressure vessels and there is scope in studying the opening in thick pressure vessel, from above discussion it is cleared that study of the effect of change in size, position, location of the opening in pressure vessel to study the stress concentration is essential, the position and location of the opening on cylinder is not studied in past by researcher and there is no code provision for such design, here PVEleite software is used for designing of pressure vessel, SAIS program also used for reducing time for calculation, Finite element analysis is an extremely powerful tool for pressure vessel. A structural analysis of the high pressure vessel will be implemented. The maximum load on a saddle may be conservative or liberal, depending upon the value of the ratio A/L used. Furthermore, the design of the saddle structure,

Stress concentration is one of the important factors to be studied in the pressure vessel opening, A review of the literature related to the stress concentration at opening in pressure vessel is presented, also the effect of the end cover on the position and size of the opening needs to be studied.

REFERENCES:

(1) M. Jadav Hyder, M Asif, "Optimization of Location and Size of Opening In A Pressure Vessel Cylinder Using ANSYS". Engineering Failure Analysis .Pp 1-19, 2008.
(2) Joship Kacmarcik, Nedelijko Vukojevic, "Comparison Of Design Method For Opening In Cylindrical Shells Under Internal Pressure Reinforced By Flush (Set-On) Nozzles". 2011
(3) 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, Vol.11, No.4, Pp.965-979, 2006
(4) J. Fang, Q.H. Tang, Z.F.Sanga, "Comparative Study of Usefulness for Pad Reinforcement in Cylindrical Vessels under External Load on Nozzle". International Journal Of Pressure Vessel And Piping 86,Pp 273-279, 2009
(5) Pravin Narale, P.S. Kachare , "Structural Analysis Of Nozzle Attachment On Pressure Vessel Design," International Journal Of Engineering Research And Application,Vol.2,Pp 1353-1358 , 2012
(6) James J. Xu, Benedict C. Sun, Bernard Koplik, "Local Pressure Stress On Lateral Pipe-Nozzle With Various Angles Of Intersection," Nuclear Engineering And Design 199, Pp 335-340, 2000
(7) Arman Ayobstress , "Analysis Of Torispherical Shell With Radial Nozzle", The Institution Of Engineers, Malaysia, Vol. 67, 2006
(8) V.N. Skopinsky, "Stress In Ellipsoidal Pressure Vessel Heads With Noncentral Nozzle," Nuclear Engineering And Design 198, Pp 317-323, 2000
(9) Jaroslav Mackerle , "Finite Element In The Analysis Of Pressure Vessels And Piping, An Addendum: A Bibliography(2001-2004)," International Journal Of Pressure Vessel And Piping 82, Pp 571-592, 2005
(10) P. Balicevic, D.Kozak, D. Kralievic, "Analytical and Numerical Solution of Internal Forces by Cylindrical Pressure Vessel with Semi-Elliptical Heads". First Serbian Congress On Theoretical And Applied Mechanics Kopaonik, Pp 10-13, 2007
(11) M. F. Hsieh, D.G. Moffat, J. Mistry, "Nozzle In The Knuckle Region Of Torispherical Head: Limit Load Interaction Under Combined Pressure And Piping Loads", International Journal Of Pressure Vessel And Piping 77,Pp 807-815, 2000
(12) 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 ,Vol I , Pp 228-234,(2012)
(13) Shaik Abdul Lathuef, Chandra Sekhar, "Design and Structural Analysis Of Pressure Vessel Due To Change Of Nozzle Location And Shell Thickness", International Journal of Advanced Engineering Research and Studies, Vol. I, Pp 218-221,2012
(14) Binesh P Vyas, R M Tayade, "Design Of Vertical Pressure Vessel Using Pvelite Software," International Journal Of Engineering Research And Technology, Vol. 2, Issue 3.
(15) Drazan Kozak, Ivan Samardzic, "Stress Analysis Of Cylindrical Vessel With Changeable Head Geometry," Science Bulletin, Serie C, Vol. XXIII

(16) Yogesh Borse, Avadesh Sharma, "Modeling Of Pressure Vessel With Different End Connections Using Pro-Mechanica", International Journal Of Engineering Research And Application, Vol. 2,Pp 1493-1497
(17) Vikram V.Mane, Vinayak H.Khatawate, "Finite Element Analysis Of Ellipsoidal Head Pressure Vessel", International Journal Of Engineering Research And Application, (2012)
(18) Ugur Guven, "A Comparison On Failure Pressures Of Cylindrical Pressure Vessels", Mechanics Research Communications 34 466–471, (2007)
(19) Pavo Balicvic, Drazan Kozak, "Strength Of Pressure Vessel With Ellipsoidal Heads", Journal Of Mechanical Engineering 5410, 685-692, (2008)
(20) Shafique M.A. Khan, "stress Distributions In A Horizontal Pressure Vessel And The Saddle Supports", International Journal Of Pressure Vessels And Piping 87,Pp 239-244, , (2010)
(21) Bandarupalli Praneeth, T.B.S.Rao, "Finite Element Analysis Of Pressure Vessel And Piping," International Journal Of Engineering Trends And Technology- Volume 3 ,Issue 5, , (2012)
(22) Drazan Kozak, Ivan Samardzic, "Overloading Effect On The Carrying Capacity Of Cylindrical Tank With Torispherical Heads For The Underground Storage Of Petrol"
(23) M. Pradeep Kumar, K. Vanisree, "Design And Implementation Of Circular Cross Sectional Pressure Vessel Using Pro-E And ANSYS", International Journal Of Morden Engineering Research, Vol 3, Pp 2350-2355
(24) L. P. Zick , "Stress In Large Horizontal Cylindrical Pressure Vessel On Two Saddle Support", .The Welding Journal Research Supplement
(25) Nishant M. Tandel, Jignesh M Parmar, "A Review On Pressure Vessel Design And Analysis, - Indian Journal Of Research", Vol 3, Issue 4, 2013
(26) Modi A J, Jadav C.S, "Structural Analysis Of Different Geometry Heads For Pressure Vessel Using Ansys Multiphysiscs", The 5th International Conference On Advance In Mechanical Engineering. 2011
(27) Hardik B Nayak, R R Trivedi, "Stress Analysis Of Reactor Nozzle To Head Junction", International Conference On Current Trend In Technology, Nirma University,2011
(28) M J Mungla, K N Srinivasan, V R Iyer, "Design And Analysis Of A High Pressure Vessel Using ASME Code", Journal Of Engineering And Technology, Vol-6, 2013