Journal of Material Science and Mechanical Engineering (JMSME)
p-ISSN: 2393-9095; e-ISSN: 2393-9109; Volume 4, Issue 2; April-June, 2017 pp. 98-102

http://www.krishisanskriti.org/Publication.html

A Review on Modeling and Analysis of Pressure Vessels

Kuldeep Singh[1], Sushovan Chattarjee[2] and K.M. Pandey[3]

[1]*M.tech. scholar, Department of Mechanical Engineering, NIT Silchar*
[2,3]*Department of Mechanical Engineering, NIT Silchar*
E-mail: [1]kuldeep6891@gmail.com, [2]sushovan@mech.nits.ac.in, [3]kmpandey2001@yahoo.com

Abstract—*Pressure vessels, boilers or steam generators with riveted joints or welded joints are the closed vessels and generally used for storing pressurized fluids in the severe pressure and temperature conditions. These vessels generate steam when heat energy supplied to water and also used in the industrial and in household applications. During pressurized fluid storage or the steam generation process, vessels are subjected to severe thermal and structural loads, which make the selection of vessels more critical. Generally, pressurized fluid or gas storing vessels are made of uniform thickness and known by the maximum circumferential stresses due to internal pressure, but the stress along the length is 1⁄2 of the circumferential stress So, the longitudinal joints makes more strong than circumferential joints. Pressure vessels and boilers with welded joints or riveted joints should be fabricated and designed as per ASME standards of the design of pressure vessels. It's necessary to design these vessels to sustain such types of load. But at the recent time, due to advancement in technology, CAD and FEA tools are used for modeling and analysis purpose of pressure vessels.*

.

Keywords: FEA, finite element modeling, ASME codes and standards, ANSYS

1. INTRODUCTION

Universally Pressure vessels have been used in power plants, nuclear power sectors and military as well as in petroleum industries for many years. These vessels contains fluids (liquid or gas) which are subjected to high pressure, high thermal stresses and various types of loads which may be in static or dynamic conditions during services. Various factors viz. the shape, material used, chemical composition, environment conditions of vessels and physical substances etc. effects vessels and are used while designing the pressure vessels. Each of the factors has different effects on the performance of the pressure vessels. The fluid being stored may undergo a change of state inside the pressure vessel as in the case of steam boiler or it may combine with other reagents as in chemical plants. So, the pressure vessels should be designed with utmost care as per ASME or IBR norms and specifications otherwise rupture of the pressure vessels means an explosion which may cause loss of life as property.

The material of the pressure vessels may be a brittle material e.g. cast iron or a ductile material e.g. mild steel or a composite material e.g. CRPF. Cylindrical or spherical pressure vessels e.g. boilers, tanks, pipes, hydraulic cylindrical etc. are commonly used in the different industries to carry both liquids and gases under pressure. Under such situation, pressure vessels are subjected various types of stresses in all the direction. The basic requirements while designing of such types of vessels are safety, reliability and economy. However the pressure vessels may work under the high pressure and high temperature. Two type of analysis method are commonly applied to pressure vessels. The first one is based on thin pressure vessel theory and also based on simple mechanics approach, defined as the ratio of inner radius (r) to wall thickness (t) or $r/t < 10$. The second one is based on the elasticity solution and always applicable regardless of r/t ratio and can also be used to find the solutions for thick pressure vessels. Finite element analysis (FEA) is a practical tool in the study of the pressure vessels, especially for determining stresses in local areas such O' rings groove cavity and other areas which is difficult to analyze manually.

2. LITERATURE REVIEW

Farah et al.[1] developed software for automating the design analysis of rivets for boiler shells. The aim of development of this software was to present a method of effective use of general purpose programs in the design analysis. The software developed is use for evaluating the effect of design parameters on rivet of boiler shell characteristics. The program is composed of user interface, which is composed in visual basic 5.0 with access database for design database for rivets, include the form for data and result output procedures.

This method is used as a powerful tool for design and analysis of rivets for pressure vessels. The user starts the application by specifying his input data as inner diameter of boiler, steam pressure, tensile stress, crushing stress, shear stress. With the help of database, the output results like thickness of shell, rivet diameter, rivets hole diameter, thickness of cover plate, margin, sharing and crushing strength of joint, tearing strength of plate with hole, tensile strength of plate without hole, efficiency are calculated.

Heckman David [2] studied the FEA result of pressurized vessel. At earlier time, primary analysis had been done by using hand calculations and empirical curves. But at present time, new computer advances have made finite element analysis a practical tool in study of pressure vessels. In this research paper analysis is had been made on a three dimensional, symmetric and asymmetric models, than the primary conclusion was made that finite element analysis is the extremely powerful tool for the analysis purposes when used in a correct manner. Depending upon the desired solution, there are different methods for faster run time and less error. The researcher suggested two methods that include symmetric models with shell elements and second one is asymmetric models with solid elements. In symmetric model, analysis is done on half part of the shell. Author suggested that boundary conditions are applied along the edges of the vessels to get nearly correct results.

William Barnet Le Van [3] studied the riveted joints in boiler shell and concluded that it is familiar to have the rivets hole one sixteen of an inches in diameter larger in rivet in order to allow their expansion when rivet is hot, however the difference between diameter of rivet hole and the rivet diameter should vary with size of rivet.

The creation of heads of rivets (of steel and iron) and shapes (conical or semi spherical) should not be charged by the riveting process. The creation of rivet head intended to be permanent and this permanent formation can only be retained by the use of “hold-fast”, which agrees the shape of head.

Kale et al. [4] studied the riveted joints with the use of adhesive using the Finite Element Analysis method. This work involved the appropriate design and characterization of such type of joints for extreme utilization. This study involves the usefulness of bend line thickness and bonded layer configuration. This work shows that riveted bonded joints seem to toughen and balance the stress distribution evenly. His work shows that when adhesive material are used, enhance the efficiency and life time of these type of joints and it is also suitable to different thickness and dissimilar metals.

Nidhi et al. [5] studied pressure divination of pressure vessel with the help of finite element analysis. The main motto of the study was to suggest different type of finite element methods to find out the burst strength of pressure vessel. After finite element analysis, the researcher saw that if all of the defined design tolerances are at their minimum value, the pressure vessel is failed and this pressure is called the burst pressure. The study mainly focused on different types of parameters which affects predict the burst strength of their cylindrical pressure vessel.

Hossein et al. [6] studied the effect of riveting sequence and rivet pitch on the characteristics of riveted joints using finite element methods. There are many parameters related with it. This paper presented a study on the effect of some controllable process parameters in riveting ie. The sequencing of the riveting and distance between rivets (pitch) on the characteristics of the riveted joints and the formed rivets. In this research paper, the study is done on a one eight inch nominal diameter flat head rivet.

Elzbieta et al. [7] studied effect of technological disfigurement on residual stress field in riveted joints. The FEA of riveted structure requires a local global approach in modeling. To find out the residual stress and strain fields through the rivet hole, a local design is used and to analyze the influence of technological disfigurement on the quality of the riveted joints. this research paper, the merge parts of the joints and application of elastic plastic material model, the contact phenomena with coulomb friction were taken into consideration

The researcher proposed a method for calculates the stress-strain curve of rivet shank diameter under compression that takes into consideration the friction between the rivet and the tool. With respect to the manufacturing process, an application of redesigning the non uniform stress distribution in the rivet hole was proposed and analyzed.

studied the effect of variation in the riveting process on the quality of riveted joints in correct selection or variation in these parameters could induce excessive residual stresses resulting in stress concentration that initiates crack. It may also result in deformation of rivet head leading to lose rivets. The research was performed on a 1/8" (0.125") diameter rivet and a 0.064" thick aluminum sheet. The results indicate that due to variation in process parameters for riveting using counter sunk rivet having depth of 0.042" for rivet hole, the rivets do not meet quality requirements. The primary reason for such situation is presence of a gap between the rivet and the hole. On reducing the counter sunk depth to 0.032", tolerances of hole and rivet diameter increases, as well as range of squeeze force also increases.

Gutman et al. [8] studied suitability of thin wall high pressure vessels under corrosion. The researcher proposed a method to determine the critical time of loss in stability of thin wall high pressure vessels prone to uniform corrosion on the inside surface. This method is established on proposed thin elastic cylindrical shell model.

Masayuki et al. [9] Investigated failure pressure of straight pipe having thinning of wall subjected to internal pressure. FEM analysis has been done to analyze the burst pressure. Three types of materials have been used: steel, stainless steel and carbon steel. The failure pressure obtained using line pipe steel was the lowest under the same flaw condition but in case different flaw conditions, the failure pressure was almost the same for carbon steel and steel. This failure pressure was lower than stainless steel under varying flaw conditions. This concludes that the assumed criteria for steel pipe can be applied to make a modest evaluation of carbon steel and stainless steel.

Bock N. and Zemen J. N. [10] discussed about the angular misalignment at longitudinal weld joints. They proposed various types of formulas for the bending stresses and local peaking of thin wall pressure vessel, based on second order theory effects. They presented a (power serious) approximation for the local peaking and compared it with the numerical results, which is obtained via FEM analysis.

Suresh et al. [11] performed analysis of bonded single riveted lap joints. In this paper, the desirable configuration and characterization of the bonded lap joints have been used for ultimate utilization. These riveted joints have effective and thick bond line and bonded layer configuration. This assumption is also suitable for dissimilar metal thickness joints. In this work, adhesives have been used at different places of riveted joints. FEM has been used to perform the analysis of joints. This analysis shows that bonded riveted joints are superior in strength to simple riveted joints. This is because of uniform stress distribution in bonded riveted joints. Thus the life span and efficiency of riveted joints is increased. For dissimilar metals, joints are free from any type of corrosion effects.

Brabin at al. [12] used FEM to obtain the peak elastic stress distribution at circumferential joint of the cylinder pressure vessel. Three types of joints, viz. un-filleted joint with uniform thickness, un-filleted joint with non-uniform thickness, filleted joint of uniform thickness have been used. The peak stress values for the considered three joints, obtained from FEM analysis are very near to the experimental results. FEM analysis showed the value of peak stress to be reduced for filleted butt joint. This paper concluded that discontinuities are always present in all metallic pressure vessels and it is always desirable to reduce such anomalities.

Khan Shafique M. A. [13] performed FEM analysis of a three dimensional model of horizontal pressure vessel along with saddle supports and analyzed their stress distribution. For saddle supports, the stress distribution has been analyzed for web, flange, and base plate. The effect of varied loading conditions and various other parameters is also investigated. For the cases of pressure vessel and saddle structure, various recommendations have been made for optimal value of the ratio of the distance of supports from the vessel end to the length of the vessel and ratio of the vessel length to the radius of the vessel for minimum stresses.

Deolia Puneet and sheikh Firoza [14] studied about the pressure at which vessel cracks and internal fluid leaks, known as a burst pressure. In this paper FEM has been used for predicting burst pressure on the basis of Ramberg-Osgood equation and then the obtained result is compared with elasto-plasto curve and true stress-strain curve. Ramberg-Oswood model has been found to show better correlation with the experimental results when compared to the modified Faupel formula.

Carbonari et al [15] discussed about shape optimization of asymmetric pressure vessels. The researcher considered an integrated approach of pressure vessel model in association with a multi objective function in order to minimize the von-misses mechanical stress from nozzle to head. For this purpose, a proper multiple objective function based on a logarithmic of a p-root of summation of p-exponent terms has been defined.
In this paper some example are examined and solution obtained for entire vessel by considering temperature and pressure loadings. For vessels with mechanical loading only and initial shapes close to a semi sphere, the maximum head stresses are similar than the cylinder stresses. Beyond this as the number of spline knots increases, the stress value also increases and also the stress value in the junction between head and cylinder decrease and the convergence stability is improved either with or without a thermal gradient.

Jaykumar M. and Christopher T. [16] studied 2-D FEM analysis with asymmetric model has been carried out using ANSYS to analyze the failure pressure of cylindrical pressure vessels made of ASTM A36 carbon steel having induced residual stress by welding. Then an elasto plastic analysis is preformed to find out the failure pressure of the pressure vessel without residual stress. After that another elasto plastic analysis is performed to access the effect of residual stress on failure pressure of pressure vessel with residual stresses. From this analysis, a reduction in failure pressure due to unavoidable residual stresses has been observed.

Zhang et al [17] proposed mathematical model for elastic deformation, plastic deformation and spring back during riveting process. Simulation of this model has been performed on ABAQUS to get more accurate results for rivet and two thin walled sheet metal parts of aluminum. Then the experimental research is compared with the simulation results.

Blanchot V. and Daidie A. [18] proposed an approach to reduce the influence of strain and residual stress on the link in the post riveting. The researchers have performed the

simulation of riveting process and analyzed its influence on the riveted link behavior.
The analysis shows that plastic strain plays no significant effect on plates and is heterogeneous in the rivet. The radial pretension has also been observed in this study. This paper concludes with the study of behavior diagram according to different load cases such as tensile loads, shearing loads or combination of two.

Deng et al [19] designed a model and created an electric and magnetic field line on this model and coupled them with the help of ANSYS. Due to these electromagnetic field lines, electromagnetic force field is created and that deform the model. When this deformed model is simulate by FEM analysis. This showed that the deformation during the rivet processing, rivets are free from upsetting and local free upsetting. Same work is also performed experimentally and experimentally result showed that the grain elongation is present in the specimen but did not show any crack in the shear around the rivet head.

Skorupa et al [20] studied experimentally and analytically results to evaluate of the secondary bending of lap joints with rivets of the aircraft sheet along the longitudinal direction. The experimental work comprised with strain gauge measurement and that measure the secondary bending stresses. fatigue test has been also carried out to recognize fatigue life of riveted joint due to the of secondary bending .Schijve model used for this purpose work on the theory of elastic bending of beams and used to evaluate the secondary bending moments in the critical joint location of riveted lap joint. Fatigue test on riveted lap joints specimen showed that variable associated with the joint geometry such as sheet thickness, rivet row spacing as well as the staggering of the sheet thickness in the overlap region can influenced on the fatigue. Schijve analytical model is used as a conventional tool to compute the secondary body moment at any location of the lap joint.

Gunay Durmus and Aydemi Ralpay [21] studied the stress distribution in the adhesive butt joint of the cylindrical pressure vessel and are analyzed by the finite element method. For this analysis purpose eight-node-quadrilateral axis symmetric finite element is used. The structure consists of two cylindrical parts under internally subjected pressure are joined together using adhesive. The researchers studied the stress distribution and the effects of the adhesive thickness on the distribution of stress in the riveted lap joint

Cangran et al [22] dealt with the riveted joint modeling for numerical analysis of air frame crashworthiness and consider three situations, first one is based on the structural embrittlement due to the riveting process, second one on mechanical strength characterization and third one is based on simplified modeling of bodings. Finite element modeling was done for each type to saw that at what extend experimental approach is replaced with the finite element simulation.
In first situation, structural embrittlement due to riveting process was identified and modeled correctly using the FE code 'Pam-Crash'. In second situation, Gurson damage parameters was identified for both aluminum alloys (rivet and sheet metal plate materials In third situation, the Arcan test setup was adapted for riveted joints and tests were performed to characterize a macroscopic failure criterion under mixed mode loading and this criterion under mixed mode loading and this criterion are also used to define the macroscopic failure criterion.

Szymczyk et al [23] investigated the FEM analysis of residual stress fields in the riveted joints and the estimation of the internal stress magnitude realizing by partial and removing of the rivet material. Deformation around the rivet hole due to stress relieving can be measured and compared with the deformation strain fields. Numerical FEA simulations of the upsetting process are carried out and contact friction between the mating parts of the joints is defined. Non destructive testing methods are used in combination with numerical calculations and showed that Radial deformation tends to increases during head removal and decreases after entire rivet removal.

Balbudhe S W AND Zaveri S R [24] dealt with the FEM analysis of the riveted lap joints for the purpose of suitable configuration and characterization for utmost utilization.FEA is performed on the riveted lap joint model and evaluate the stress and fracture analysis results under the residual stress field and external loading under tension and also performed two step FEM simulation for riveting process under tension loading. Then evaluate the residual and overall stress. After analysis the results showed the semielliptical surface crack at flying surface of plates and quarter elliptical mark at the rivet hole and then examined these cracks.

3. CONCLUSION

Finite Element Analysis is an extremely powerful tool for boiler shell or pressure vessel analysis when used correctly and it would be expected that students of engineering as well as design engineers use CAE software as a powerful tool for modeling and analysis purpose of any structures with riveted or bolted or welded joints of the lap or butt types.

REFERENCES

[1] Farah KamilAbid Muslim, Dr.Essan L. Esmail, “Computer Aided Design of rivets for steam boiler shell,” Al- Qadisiya Journal for Engineering Sciences, Jan 2012 pp.377-393.

[2] David Heckman, "Finite Element Analysis of Pressure Vessels," University of California, Davis MenterCreneMassion, Mark Greise,Summer 1998, pp.1-7.

[3] Willaim Barnet Le Van, "Riveted joints in boiler shell," Read at the state meeting of institute, Nov 19,1890.

[4] NidhiDwivedi, Veerendra Kumar, "Burst Pressure Prediction of pressure vessel using FEA," International Journal of Engineering Research and Technology (IJERT), September 2012, pp.1-5.

[5] Kale Suresh,K.L.N. Murty and Jaynanda Kumar, "Analysis of Adhesively Bonded Single lap riveted joint using ANSYS," International Journal of Mechanical and Industrial Engineering (IJMIE),Jan 2012, pp.77-82.

[6] Faisal Aman and S. HosseinCheraghi and Krishna K. Krishnan and Hamid Lankarani, "Study of the impact of riveting sequences, rivet pitch and gap between sheets on the quality of riveted lap joints using finite element method," springer-verlag London Limited,Oct. 2012, pp.545-562.

[7] S. HosseinCheraghi, "Effect of variations in the riveting process on the quality of riveted joints," springer-verlagLondan Limited, DEC.2008, pp.1144-1145.

[8] ElzbietaSzymczyk, Jerzy Jachimowich, GrezegorzSlowinski, AgnieszkaDerewonkoa, "Influence of technological imperfactions on residual stress fields in riveted joints," Procedia Engineering, April 2009, pp.59-62.

[9] E.Gotman, J. Hadded, R.Bergman, "Stability of thin walled high pressure vessel subjected to uniform corrosion," Thin walled structures,April 2000 pp. 43-52.

[10] Masayuki Kamaya,TomohisaSuzuke, Toshiyuki Meshi," Failure pressure of straight pipe with wall thinning under internal pressure,"International Journal of Pressure Vessels and Piping, Nov. 2007, pp. 628-638, Nov. 2007.

[11] Bock N., Zeman J. L., "On Bending Stresses at Longitudinal Weld Joints of Cylindrical Shells Due to Pecking," International Journal of Pressure Vessels and Piping, June 1993 pp.103-106.

[12] T. AseerBrabin, T Christopher, B. Nageswara Rao, "Finite Element Analysis of cylindrical pressure vessel having a misalignment in circumferential joint," International Journal of Pressure Vessels and Piping, May 2010, pp.197-201.

[13] Safique M. A. Khan, "Stress distribution in a horizontal pressure vessel and the saddle supports,"May 2010 pp.239-244.

[14] DeoliaPuneet, Shaikh Firoz A., " Finite Element Analysis to estimate burst pressure of mild steel pressure vessel using Ramberg-Osgood model," Perspective in Science, Sept.2016, pp. 733-735.

[15] R. C. Carnobari, P. A. Munoz-Rojas, E. Q. Andrade, G. H. Paulino, K. Nishimoto, E.C. N. Silva,"Design of pressure vessels using shape optimization:An integrated approach," International Journal of Pressure Vessels and Piping, May- July 2011, pp.198-212.

[16] Jaykumar M., Christopher T., "Influence of residual stresses on failure pressure of cylindrical pressure vessels,"Chinese Journal of Aeronautics, Dec. 2013 pp.1415-1421.

[17] ZHANG Kaifu, CHENG Hui, YUAN Li, "Riveting process modeling and simulation for deformation analysis of aircraft's thin walled shell- metal parts," Chinese Journal of Aeronautics, June 2011, pp.369-377.

[18] Blan Chat V., Daide A., "Riveted assembly modeling:study and numerical characterization of riveting process," Journal of processing and materials, Dec. 2006, pp.201-209.

[19] Deng J. H.,H.P. Yu, C.F.Li, "Numerical and a experimental investigation of electromagnetic riveting," Material Science and Engineering, Jan. 2009 pp.242-247.

[20] M Skorupa, A Korpel, A Skorupa, T. Machaniewicz, "Observations and analysis of secondary bending for riveted lap joints," International Journal of Fatigue, March 2015 pp.1-10.

[21] Durmus GUNAY, Alpaya AYADEMIR, "Stress analysis in Adhesive butt joints of cylindrical pressure vessels," Mathematics and Computational Applications, Jan.1999,pp.233-239.

[22] B.Langrand, E Deletose, E. Markiwicz, P. Drazetic, " Riveted joint modeling for numerical analysis of airframe crashworthiness," Finite Element Analysis and Design, Dec.2001, pp. 21-44.

[23] ElzbietaSzymczyk, AgweszkaDerewonka, GrzegorzSlawinski, Jerzy Jachimowicz, "Numerical Estimation of internal stress relieving in destructive test," Computer Assisted Mechanics and Engineering Science, Jan. 2010, pp.185-192.

[24] Suyogkumar W Balbudhe, S. R. Zaveri, "Stress analysis of riveted lap joint," International Journal of Mechanical engineering and Robotic Research (IJMERR), July 2013 pp.137-143.