Burst Pressure Prediction of Pressure Vessel using FEA

Nidhi Dwivedi,

Research Scholar (G.E.C, Jabalpur, M.P),

Veerendra Kumar

Principal (G.E.C, Jabalpur, M.P)

Abstract

The main objective of this paper is to propose various types of Finite Element Methods used for the calculation of burst strength of pressure vessel. The pressure at which the pressure vessel should burst if all of the specified design tolerances are at their minimum values is called burst pressure. Prediction of burst strength is the very important aspect in the pressure vessel design. The present study mainly focuses on various types of factors which tremendously affect the burst strength of pressure vessel. FEA is a very powerful tool used to determine burst strength of pressure vessel. Axi-symmetric FEA is carried out to accurately predict the burst strength of a thin cylindrical pressure vessel.

Keywords: *Burst Pressure, Axi-symmetric model, elliptical end caps*

Nomenclatures

D = Mean Diameter of Vessel

D_i = Internal Diameter of Vessel

D_o = Outer Diameter of Vessel

d_o = Outer Diameter of Nozzle

d = Mean Diameter of Nozzle

L_v = Length of Vessel

L_n = Length of Nozzle

P_b = Burst Pressure

R = Mean Radius of Vessel

R_i = Internal Radius of Vessel

R_o = Outer Radius of Vessel

T = Thickness of Vessel

t = Thickness of Nozzle

σ_u = Ultimate Tensile Strength of Vessel

σ_y = Yield Strength of Vessel

Introduction

The pressure vessels are used to store fluids under pressure. The material of pressure vessel may be brittle such as cast iron, or ductile such as mild steel. According to the dimensions, the pressure vessels can be classified as thick shell or thin shell. If the wall thickness of the pressure vessel is less than 1/10 nth of the diameter of the shell, then it is called thin shell pressure vessel and if the wall thickness of the shell is greater than 1/10 nth of the diameter of the shell, then it is called as thick shell pressure vessel. This paper mainly deals with the study of thin cylindrical pressure vessels. Failure of thin cylindrical pressure vessel occurs in two ways. It may fail along the longitudinal section i.e. circumferentially or it may fail along the transverse section i.e. longitudinally. Two types of tensile stresses occur in pressure vessels. One is circumferential or hoop stress and the other one is longitudinal stress. Longitudinal stress is half of the circumferential or hoop stress. Therefore, the design of the pressure vessel must be based on the circumferential or hoop stress. Various researches were being carried out to find out the best method which can accurately predict the burst pressure of cylindrical shells. Various formulas, theories and methods are being developed to find out the exact value of the burst pressure and the exact location of failure.

Fig: 1 Industrial Pressure Vessel

Literature Review

Finite element analysis is used to predict the failure conditions of pressure vessels. Local areas such as penetrations, o-ring grooves are considered. Three-dimensional symmetric and axi-symmetric models are studied using Finite Element (FE) tool. It has faster runtime and less error. Both shell and solid elements are employed in the analysis. Problems such as local stress risers, unrealistic displacements are investigated. Linear (SHELL 63) and quadratic (SHELL 91) elements are used. Eight-node brick (SOLID 45) elements are used to compare with other elements. Linear (PLANE 42) and quadratic (PLANE 82) elements are used. Point to point contact elements (CONTAC 52) are also used [[1]]. The non linear Finite Element Analysis (FEA) is carried out to predict the plastic collapse load of pressure vessel, considering two cases i.e.; with and without defects. Iterative techniques are adopted to find out the failure pressure. The total time consumed is minimized and the total cost estimated is reduced in the present study [[10]]. The large displacements and plastic straining response of the structure are considered for the analysis. Non-linear FEA is carried out. The limit load predictions of imperfect tubes having ovalized cross sectional shape under external pressure is done. The large strain cold deforming process of a pressure vessel is also studied. Iso-parametric, eight-node, two dimensional solid elements are used [[12]]. Two parameters were considered i.e.; limit pressure and Stress Concentration Factor (SCF). Three dimensional FEA is carried out to obtain better results. The local membrane stress criterion has been considered. Twenty-node solid iso-parametric elements (SOLID 95) of ANSYS software are employed [[11]]. The comparative study is carried out between the existing formula mentioned in European standard and the FEA results. Two types of ductile material are used, P 355 steel alloy and P 500 QT steel alloy. The cylinder to nozzle intersection is studied. Combination of three-dimensional solid modeling with three- dimensional shell modeling is done to obtain good results [[9]]. Both the linear elastic and elastic- plastic stress analysis is carried out to study the cylindrical shell intersection. The limit load and the burst pressure are calculated after finding out the stress concentration and flexibility factor. Two types of methods, Double Elastic Slope Method and Tangent Intersection Method are employed to calculate the burst pressure. The Arc Length method is employed to determine the failure location in the pressure vessel. The validation of FEA result is done with the test results. Three-dimensional twenty node structural solid elements are used to generate the model mesh [[2]]. FEA is carried out on pressure vessel. Defect geometry and loading conditions are considered. New analytical local and global collapse loads are derived. The global collapse load gives good agreement with the FEA results. Two types of defects are studied i.e.; semi-elliptical surface defect and infinitely long defect in a pipe [[5]]. Finite Element Analysis is carried out using ANSYS software. Three dimensional twenty node structural elements are employed to perform a static non linear analysis of pressure vessel. FEA gives good agreement with the test results [[4]]. The concept of wall thinning is investigated and three-dimensional elastic plastic Finite Element Analysis is carried out. Three types of materials: line pipe steel, carbon steel, and stainless steel are studied. The pipe is modelled by using eight-node solid elements [[6]]. FEA is carried out to accurately predict the burst pressure of cylindrical vessels. A static non linear analysis is carried out using three-dimensional twenty node structural solid elements. The Newton-Raphson Method and the Arc Length Method are used. Barlow equation is also investigated and is found to give good agreement with the FEA results [[3]]. FEA is carried out using ANSYS software. Arc Length Method, Newton-Raphson Method and Double Elastic Slope Method are employed to obtain the better results. The material of the vessel is assumed to be elastic perfectly plastic. The parametric FEA is performed using the ANSYS-APDL software. Three-dimensional eight node solid element SOLID45 is adopted to mesh the structure [[7]]. FEA has been carried out to obtain the elastic stress distribution at cylinder to cylinder junction in pressurized shell structures. Three joint configurations are used i.e.; un-filleted butt joint with equal thickness, un-filleted butt joint with unequal thickness and filleted butt joint with equal thickness. The peak stress value is found to reduce due to filleted butt joint. An axi-symmetric model with element PLANE 42 has been used for the analysis [[8]].

Dimensions of Cylindrical Shell

A test vessel was designed and fabricated for the experimental study. It consists of a main vessel, a nozzle, two elliptical heads, etc. It should be noted that the dimensions of the test vessel were determined by a Pressure Vessel Research Council (PVRC) oversight committee. Fig: 2 illustrates the configuration and geometric dimensions of the test vessel [[2]].
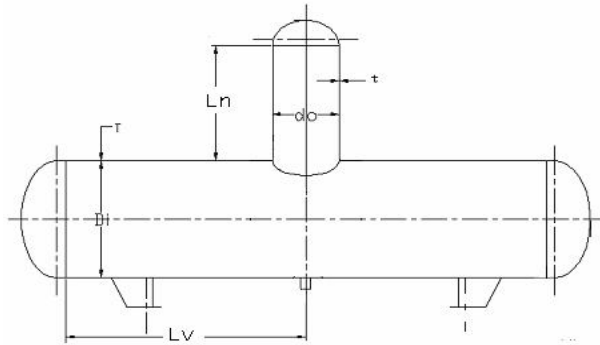
Fig: 2 Geometric Dimensions of Test Vessel

Table: 1 Dimension of test vessel

Di in mm	600
T in mm	6
do in mm	325
t in mm	6
L_v in mm	1200
L_n in mm	600
d/D	0.526
t/T	1.0
D/T	101

Material Properties

The material of the main vessel and nozzle is Q235-A (low carbon steel). This material has an elastic tensile modulus of 2.01×10^5 MPa which is used throughout the analysis. Average values of the yield strength σ_y and ultimate strength σ_u for material Q235-A are 339.4 MPa and 472 MPa, respectively. Poisson's ratio is taken as 0.3 for this material [[2]].

Experimental Result used for Validation

The test vessel was pressurized in small increments to burst. The burst pressure is 7.4 MPa. The failure occurs at the intersection area of the vessel and nozzle [[2]].

Methodology

Finite element static, non-linear analysis of the model vessel has been performed using ANSYS software. Due to the symmetry about the longitudinal and transverse plane, only quarter part of the vessel is analyzed. For the internal pressure load case, symmetry boundary conditions are employed on the two symmetry planes.

The boundary condition used in the analysis is that, all the nodes on the symmetric section i.e. longitudinal plane and transverse plane are constrained against deformation. Further the left end of the vessel is fixed while the right end of the vessel and the end of the nozzle are free. The yielding is based on the Von-Mises Yield criterion. The material is assumed to be isotropic.
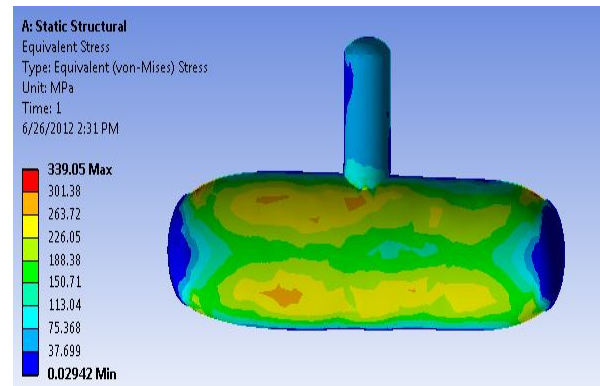
(a) Half vessel with end caps
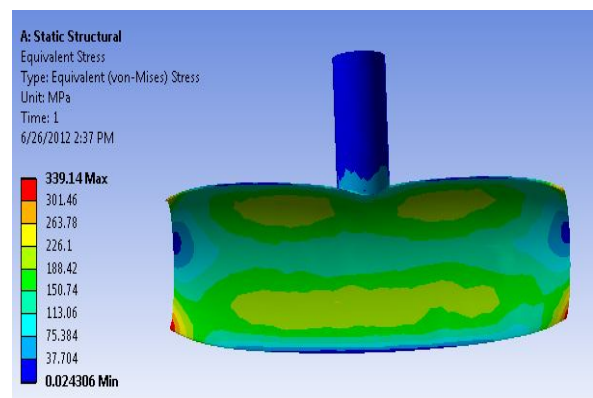
(b) Half vessel without end caps

Fig: 3 FEA of Half Vessel

Results and Discussion

Two cases are considered that is vessel with end caps and vessel without end caps. The effect of end caps is high. Considering the Von Mises Yield criteria, when vessel with end caps is analyzed, the burst pressure came out to be 6.47 MPa. When the vessel without end caps is analyzed, then the burst pressure came out to be 4.85 MPa. In former case the relative error is 12.567% lower

than the experimental value, and in later case the relative error is 34.459% lower than the experimental value. It is analyzed that the vessel having end caps gave much better results than the vessel without end caps.

Table: 2 Results of FEA

	Half vessel with end caps	Half vessel without end caps
Pressure	6.47 MPa	4.85 MPa
Total deformation	0.732 mm	0.6654 mm
Equivalent stress	339.05 MPa	339.14 MPa
FOS	1.001	1.0008
Relative error	12.567%	34.459%

Comparison & conclusion

It is observed that the relative error between the experimental value and the FEA result in case of the half vessel with end caps is much better than the half vessel without end caps. It is suggested to analyze vessel along with their end caps to obtain better results.
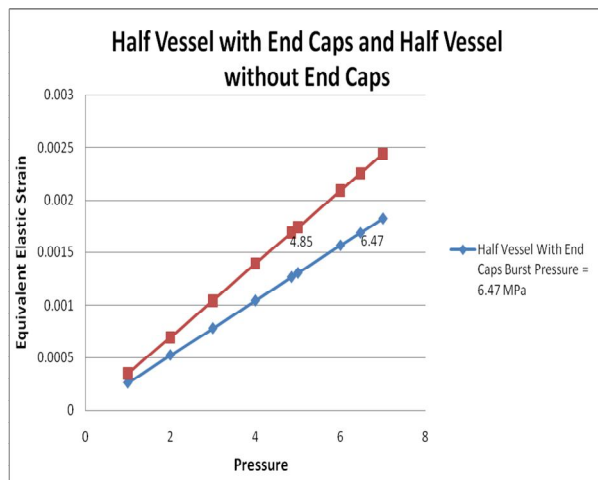

Fig: 4 Equivalent Elastic Strain versus Pressure

References

[1] David Heckman and Davis, Finite Element Analysis of Pressure Vessel, *MBARI* (1998); 1-7.

[2] Liping Xue, G.E.O Widera and Zhifu Sang, Application of FEM Analysis Methods to a Cylinder-Cylinder Intersection Structure, *18th International Conference on Structural Mechanics in Reactor Technology SMIRT 18* (2005); F07: 4.

[3] Liping Xue, G.E.O. Widera and Zhifu Sang, Burst Analysis of Cylindrical Shells, *Journal of Pressure Vessel Technology* (2008); 130: 014502 (1-5).

[4] Liping Xue, G.E.O. Widera and Zhifu Sang, Burst Pressure Prediction of Cylindrical Shell Intersection, *Transactions SMIRT 19* (2007); F01: 5.

[5] M. Staat, Local and Global Collapse Pressure of Longitudinally Flawed Pipes and Cylindrical Vessels, *International Journal of Pressure Vessel and Piping* (2005); 82: 217-225.

[6] Masayuki Kamaya, Tomohisa Suzuki and Toshiyuki Meshii, Failure Pressure of Straight Pipe with Wall Thinning under Internal Pressure, *International Journal of Pressure Vessel and Piping* (2008); 85: 628-634.

[7] Peng-Fie Liu, Jin-Yang Zheng, Li Ma, Cun-Jian Miao and Lin-Lin Wu, Calculation of Plastic Collapse Load of Pressure Vessel using FEA, *Journal of Zhejiang University SCIENCE A* (2008); 9(7): 900-906.

[8] T. Aseer Brabin, T.Christopher and B.Nageshwara Rao, Finite Element Analysis of Cylindrical Pressure Vessels having a Misalignment in a Circumferential Joint, *International Journal of Pressure Vessel and Piping* (2010); 87: 197-201.

[9] Th. Diamantoudis and Th. Kermanidis, Design by Analysis versus Design by Formula of High Strength Steel Pressure Vessel: A Comparative Study, *International Journal of Pressure Vessel and Piping* (2005); 82: 43-50.

[10] W.payten and M.Law, Estimating the Plastic Collapse of Pressure Vessels using Plasticity Contours, *International Journal of Pressure Vessel and Piping* (1998); 75: 529-536.

[11] You-Hong Liu, Bing-Sheng Zhang, Ming De Xue and You-Quan Liu, Limit Pressure and Design Criterion of Cylindrical Pressure Vessels with Nozzles, *International Journal of*

Pressure Vessel and Piping (2004); 81: 619-624.

[12] Z.Sanal, Non-Linear Analysis of Pressure Vessels: Some Examples, *International Journal of Pressure Vessel and Piping* (2000); 77:705-709.