International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 04, April-2018

Stress Analysis in Compound Cylinders and Autofrettaged Cylinders

M. Kumaresan[1] & [1] Assistant Prof
Mechanical Department
SBM College of Engineering
Dindigul.

Dr. K. Chockalingam[2]
[2]Assistant Prof
Mechanical Department,
Thiagarajar College of Engineering,
Madurai.

Abstract **- Pressure vessels play vital role in aerospace, nuclear and armaments industries. Compound cylinders take advantage of withstanding higher pressures. Finite Element Analysis is a powerful analysis tool to simulate material behavior. Hence in order to understand the stress levels, failure behavior and effect of autofrettage process , an effort is made to simulate single and compound cylinders.. The results can predict the best autofrettage pressure,maximum von-misses stress across the wall of the cylinder and the optimum diametral interference which sufficiently brought the contact surface of the inner and outer cylinders to the point of yielding. To reduce the flow stress within the wall of the cylinder, the autofrettage pressure must be greater than the working pressure. For pressures lower than the working pressure the flow stress remains unchanged . The results are verified with theory and found to be very close to each other**

Keywords: Single Cylinder; Compound Cylinder; Stress Levels; Failure Analysis; Autofrettage; Finite Element

1. INTRODUCTION

Fluids are stored under pressure in pressure vessels or shells and transmitted from one place to the other through pipes. Pressure vessels are made of cast iron, sheet steel and nonferrous alloys. Special material is used for chemical vessels. Cylindrical pressure vessels are used in various technological fields such as chemical and nuclear industries, rocket motor case manufacturing and production of many weapon systems. The severity of functional performance requirements, high pressure extreme temperature leads to exact design which includes determination of extent of stress and strain, establishing the behavior of the material involved and evaluating the compatibility of these two factors. Failure due to yielding occurs when some functional of stress or strain is exceed and fracture occurs when an existing crack extends. Understanding the failure phenomenon in the cylindrical pressure vessel is complicated due to the presence of biaxial and tri-axial state of stresses and strain beyond elastic range. The determination of stress distribution in elastic plastic bodies with cylindrical or spherical symmetries has received considerable attention due to its importance in engineering applications. The prediction of failure pressure that a cylindrical pressure vessel can withstand is an important consideration in the design of pressure vessels. The mathematical theory of plasticity has been in use for many years but the formulation of a field equation for simulation of elasto-plastic deformation is still a matter of discussion. This work aims at determination of failure pressure of thin thick and compound cylinders using analytical expression and non-linear finite element analysis and comparison with published test results. Autofrettage is a well known elasto-plastic technique to increase the pressure capacity of the thick walled cylinders. In the first,the cylinder is subjected to internal pressure so that its wall becomes partially plastic. The pressure is then released and the resulting residual stresses increase the pressure capacity of the cylinder in the next loading stage. This procedure is called ‘autofrettage’. In the second technique, two or more cylinders are shrunk into each other with different diametral interferences to form a compound cylinder. The shrinkage produces a residual stress distribution within the walls of the cylinders, which improves the cylinder behavior against working pressure. This work aims at study on effect of autofrettage on single and compound cylinders.

2. THEORY OF ELASTIC STRESSES IN THIN, THICK, AND COMPOUND CYLINDERS

A cylindrical shell is considered thin if the ratio of the thickness of the shell to its diameter is less than $1/10$ to $1/15$. They find application in water pipes, liquid storage tanks, boilers and some components of airplanes, etc. Consider a thin cylindrical shell of radius r, thickness t, and length L subjected to an internal fluid pressure of p.

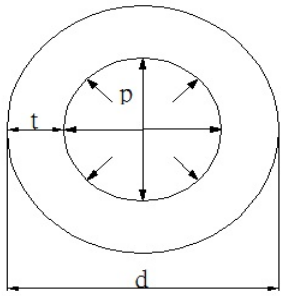

Figure (1) Geometry of a cylinder

Bursting force = pLd

If σ_h is the hoop stress due to fluid pressure, it is obtained by dividing the bursting force by the projected area of the cylinder.

Circumferential stress $\sigma_h = (pLd)/(2tL)$

$$\sigma_h = pd / 2t \qquad (1)$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 04, April-2018

If σ_m is the longitudinal stress, then tearing force is resisted by bursting force.

Here, Tearing force = $(\pi dt)\ \sigma_m$

And, Bursting force = $\left(P\frac{\pi}{4}D^2L\right)$

Equating the above forces, we get

$$\left(P\frac{\pi}{4}D^2L\right) = (\pi dt)\ \sigma_m$$

Therefore, $\sigma_m = pd/4t$ (2)

In case of thin cylinders, both hoop stress and longitudinal stress are assumed to be fairly uniform and in case of thick cylinders, the longitudinal strain is assumed to be almost uniform throughout the length. The hoop stress varies from a maximum at the inner surface to a minimum at the outer surface of the cylinder.

A hollow thick cylinder subjected to internal pressure is contributes to following stresses which are expressed by lame's equations.

Radial stress $$\sigma_r = \frac{p_1r_1^2 - p_2r_2^2}{r_2^2 - r_1^2} + \frac{r_1^2r_2^2(p_1 - p_2)}{r^2(r_2^2 - r_1^2)} \quad (3)$$

Hoop stress $$\sigma_h = \frac{p_1r_1^2 - p_2r_2^2}{r_2^2 - r_1^2} + \frac{r_1^2r_2^2(p_1 - p_2)}{r^2(r_2^2 - r_1^2)} \quad (4)$$

Longitudinal stress $$\sigma_m = \frac{p_1r_1^2 - p_2r_2^2}{r_2^2 - r_1^2} \quad (5)$$

ANSYS version-11 FEA analysis package was employed for calculating the stresses. Since the aim of the present work was to assess stress, material properties were assumed with young's modulus 75 GPa and a Poissons ratio of 0.3. One-half model of the vessel with length 70 mm , thickness 20 mm and diameter 206 mm had taken because of axi-symmetric.The meshes were prepared using preprocessor program of ansys 4 node 2-dimensional solid element (PLANE-42).

In the FEA analysis, internal pressure was applied as a distributed load to the inner surface of the FEA model. Boundary condition of the FEA model was designed closely to the working condition of the pressure vessel, the nodes at the bottom horizontal line of the vessel were constrained Y direction and the nodes at the top vertical line of the vessel were constrained in X direction. The problem had solved, the required stresses were taken from the General postprocessor. The problem had solved for various internal pressures similar to procedure explain above.

The hoop stress, longitudinal stress and radial stress were taken from the finite element. They were compared with theoretical values by using a graph.

Two or more cylinders shrunk into each other with different diametral interferences form a compound cylinder. The shrinkage produces residual stress distribution within the walls of the cylinders, which improves the cylinder behaviour against the working pressure. When two cylinders are shrunk together, the inner and outer cylinders are subjected to an internal and external pressure, respectively. Figure(2) shows a compound cylinder.

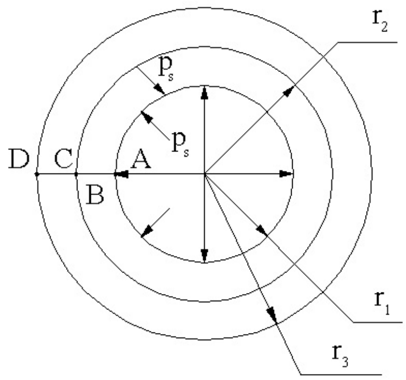

Figure (2) Compound cylinder

When two cylinders are shrunk together, a pressure is produced at the contact surface of the cylinders. This pressure for elastic shrinkage is

$$= P_s r_2\left[\frac{1}{E_1}\left(\frac{r_2^2 + r_1^2}{r_2^2 - r_1^2}\right) - \gamma_1\right] + \left[\frac{1}{E_2}\left(\frac{r_3^2 + r_2^2}{r_3^2 - r_2^2}\right) + \gamma_1\right] \quad (6)$$

Where,

Shrinkage allowance, $\delta = \delta_1 + \delta_1$

P_s = shrink fit pressure

δ = value of interference between the inner and outer cylinders

E = young's modulus

ϒ= poisson ratio

For Shrunk fit cylinders

The stresses developed in the cylinders are given by,

$$(\sigma_\theta)_A = -2P_s\frac{r_2^2}{r_2^2 - r_1^2} \quad (7)$$

$$(\sigma_\theta)_B = -P_s\frac{r_1^2 + r_2^2}{r_2^2 - r_1^2} \quad (8)$$

$$(\sigma_\theta)_c = P_s\frac{r_3^2 + r_2^2}{r_3^2 - r_2^2} \quad (9)$$

$$(\sigma_\theta)_d = 2P_s\frac{r_2^2}{r_3^2 - r_2^2} \quad (10)$$

3. FINITE ELEMENT ANALYSIS FOR STUDY OF STRESSES

An effort is made to model and analyse a thin cylinder with heavy spherical end subjected to internal pressure. An area of revolution representing wall of pressure vessel is modeled with radius measured from Y-axis. PLANE 42 element type of analysis is used with axi-symmetric option for meshing the area. Ends of the model are constrained in X and Y direction suitably. Internal pressure is applied at the inner edges. The material properties used are Young's modulus 75 GPa and Poisson's ratio 0.3 which corresponding to Aluminium alloy. The dimensions used are internal radius 103 mm and thickness 2.6 mm. The applied pressure is 5 MPa.

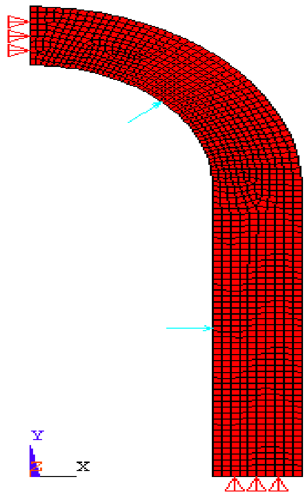

Figure (3) Axi-symmetric Finite Element Model of a closed ended cylinder

Similarly a thick cylinder of internal radius of 103 mm and thickness 50 mm is modeled and analyzed, with the same material properties. Fig(3) shows the finite element model used for both the above analysis.

In order to study the behavior of compound cylinders, the radii assumed are 50 mm and 75.25 mm for internal cylinder, 74.75 mm and 100 mm for outer cylinder. Material properties are of Alluminium alloy . Element type is, plane 42 with plane strain option. Fig(4) shows the finite element model of compound cylinder. In order to model the contact between inner and outer surface cylinders, contact 174 and Target 170 are used. X and Y constraints are applied to the relevant edges of quarter-symmetric model. It is initially analyzed with zero internal pressure to study the effect of interference.

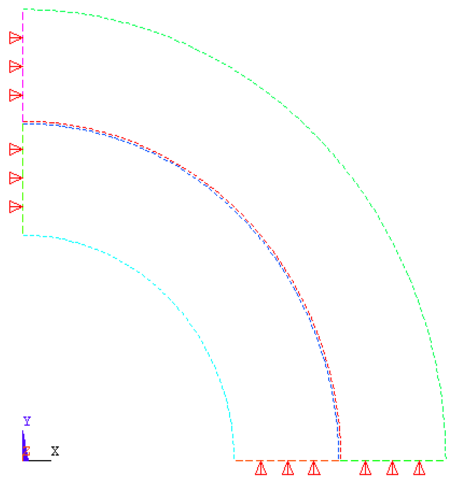

Figure (4) Plain strain model of compound cylinder

4. FAILURE ANALYSIS OF CYLINDERS

The most common testing technique adopt to obtain the material property is the uni-axial tension test. This is because this test provides basic information on strength of materials and serves acceptance test for specification of material. The data obtained from tension test are generally plotted as stress strain curve. The strength data generally provides are (i) yield strength (ii) ultimate strength and (iii) strain hardening exponent. With this available data, either a bilinear or a continuous true stress-true strain curve may be generated. Engineering material commonly used for the design of structures and pressure vessels have an initial stress-strain relation which for practical purposes, may be assumed linear, indicating that stress is directly proportional to strain and is represented by equation

$$E = \frac{\sigma}{\varepsilon} \qquad (11)$$

Elastic-plastic nature applies to materials that exhibit time independent, non linear behavior. To define the non linear deformation of material, material curve is necessary. Material curve can be generated by using the inverse Ramberg-Osgood relation

$$\sigma = E\varepsilon\,[1+ (\varepsilon/\varepsilon_0)^{n}]^{-(1/n)} \qquad (12)$$

$$\varepsilon_0 = \sigma_0/E$$

Where,

E is the Young's modulus, ε_0, σ_0, and n are material constants, obtained by getting the test data to equation (12)

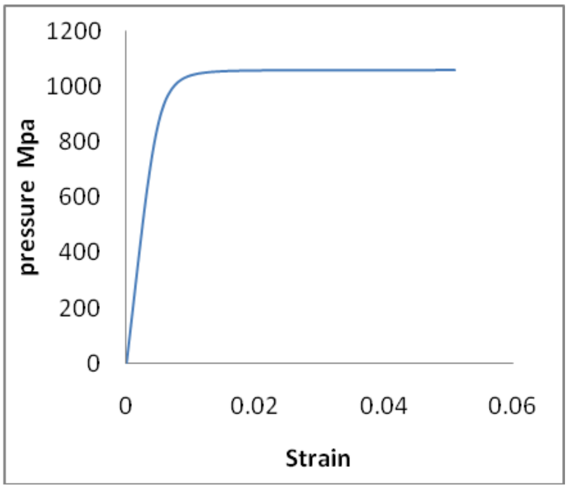

Figure (5) Stress-strain curve of Aluminium alloy

Fig (5) shows the stress-strain curve of Aluminium alloy, a high pressure pump material

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 04, April-2018

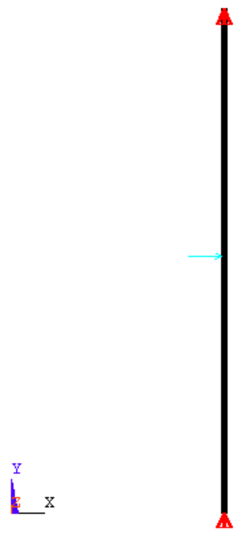

Figure (6) Finite Element Model of a thin cylinder

Table (1) shows corresponding numerical data generated using equation (12)

STRAIN	STRESS
0	0
0.001	74.956
0.004	124.1
0.005	130.5
0.008	139.7
0.009	141.29
0.014	145.44
0.015	145.88
0.016	146.27
0.017	146.59
0.018	146.87
0.019	147.12
0.02	147.33
0.021	147.52
0.022	147.69
0.023	147.84
0.024	147.98
0.025	148.1
0.026	148.21
0.027	148.31

Table (1) Numerical data of stress-strain for FEA

A close ended pressure vessel subjected to internal pressure having outer diameter 28.6 mm and thickness 2.6 mm is modeled and analysed. Finite element analysis (FEA) has been carried out here to find the possible correlation of strength parameters measured on uniaxial tensile test specimens with performance of the material in actual cylindrical pressure vessels. An axi-symmetric four node quadrilateral finite element available in ANSYS software package is utilized to model the cylindrical pressure vessel. The finite element model consists of 100 elements with 121 nodes. Axial displacement is suppressed at both ends of the cylindrical

Shell to have no axial growth under internal pressure. The true stress-strain curve is fed as material property in addition to Young's modulus and Poisson's ratio. Non-linear analysis has been carried out in the cylindrical pressure vessel with an applied internal pressure of 150MPa. Load is given by means of 300 sub steps to get the stress patterns for corresponding loads. The analysis stopped at the load step 117 indicating the failure pressure to be 58 MPa. Thus the ANSYS has the provision for checking the global plastic deformation (GPD).It indicates the pressure vessel to cause complete plastic flow through the cylinder wall (i.e., bursting pressure).Figure(8) shows variation of effective stress on outer surface and inner surface as applied internal pressure is increased in steps .It is observed that the failure occurred almost when the effective stress curves on inner surface and outer surface meet each other.

Bursting pressure of a cylinder can be calculated Using popular formula known by the researcher Faupel's

$$p_b = \frac{2}{\sqrt{3}}\sigma_{ys}\left(2 - \frac{\sigma_{ys}}{ult}\right)\ln\left(\frac{R_0}{R_i}\right) \qquad (13)$$

Similarly a thick cylinder has been designed whose outer diameter 28.6 mm and inner diameter 13 mm and length 70 mm with an internal pressure of 150 MPa. Load is given by means of 300 sub steps to get the stress patterns for corresponding loads. The analysis is stopped at the load step 270 indicating the failure pressure to be 135 MPa.

5. RESULTS AND DISCUSSIONS

In order to develop understanding of various stresses arising in the walls of thin and thick cylinders, the theoretical values obtained and compared with those obtained through finite element analysis. Table 2 presents a comparison of hoop and meridional and hoop stresses in thin cylinders.

Comparison of stresses in thin cylinder

Hoop stress(MPa)		Meridional stress(Mpa)	
Eq(1)	FEA	Eq(2)	FEA
198.08	200.6	99.04	97.805

Table (2)

Figure (7) shows a graphical comparison of hoop, meridional and radial stresses in a thick cylinder.

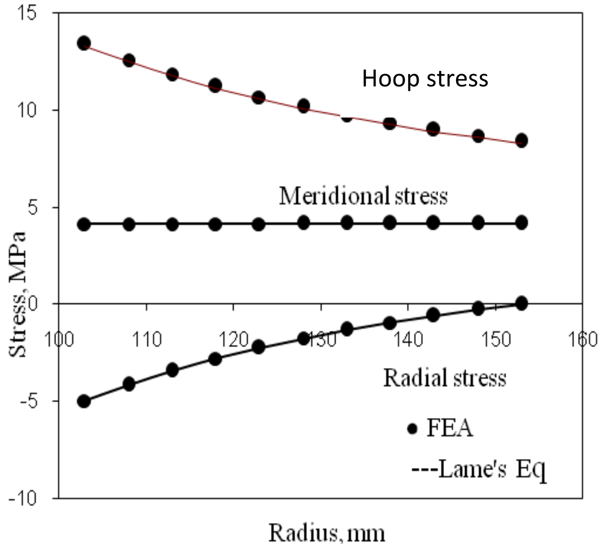

Figure (7) Comparison of stresses in thick cylinder

Table (3) and Figure (8) show the details of contact pressure in a compound cylinder

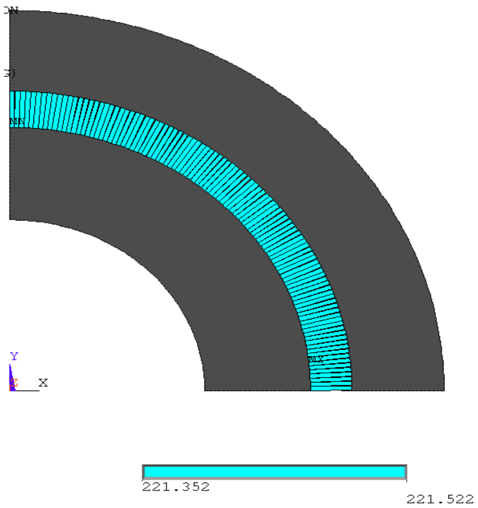

Figure (8)
Contact pressure in a compound cylinder

Comparison of contact pressure in compound cylinder

	Contact pressure Mpa
Equation	216
FEA	221

Table (3)

Table (4) shows a comparison of hoop stresses computed from equation 7-10 and obtained from FEA. From the above discussions, it can be seen that the finite element analysis results are very close to the theoretical values.

	Theory	FEA
$(\sigma_\theta)_A$	-78.465	-75.63
$(\sigma_\theta)_B$	-56.56	-53.26
$(\sigma_\theta)_C$	78.99	72.65
$(\sigma_\theta)_D$	57.09	50.05

Table (4)

Failure pressure of the thin cylinder as discussed above has been calculated using Faupel's formula (13) and compared with the value obtained from GPD check of ANSYS. Table (5) presents the comparison.

Method	MPa
Faupel's formula	56
GPD check	58

Table (5)

Figure (9) shows the variation of effective stresses in the inner surface, middle surface and outer surface of the thin cylinder as the applied pressure is increased up to the GPD. It is to be noted that all the effective stress curves meet at GPD which declares failure of the pressure vessel.

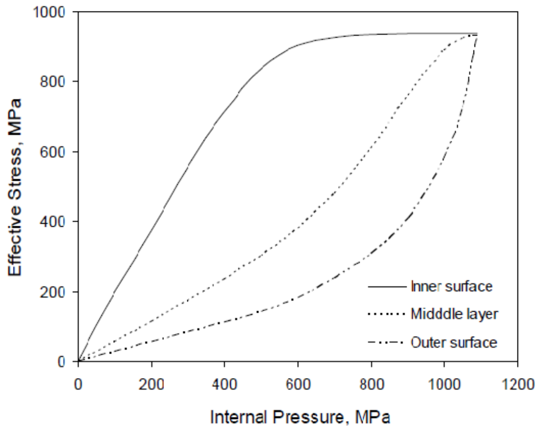

Figure (9) Effective stress Vs Applied pressure-Thin cylinder

Failure pressure of thick cylinder discussed above hasbeen calculated using Faupel's formula and compared with the failure pressure obtained using FEA Table (6)

METHOD	MPa
Faupel's Formula	131
GPD check	135

Table (6)

6. CONCLUDING REMARKS

Finite Element analysis is effectively carried out in simulation of a cylindrical pressure vessel. The stress levels in thin, thick and compound pressure vessels have been determined by FEA and compared with elastic stresses obtained by GPD check of ANSYS is found to be very close to the test failure pressure and the value obtained through Faupel's formula which indicates the validity of FEA procedure. The work is to be extended in determination of failure pressure of compound cylinders under various shrink fit pressure condition. The failure pressure of compound cylinders are significantly higher than that of a single cylinder with same geometry and material used in this work and studied the effect of autofrettage on failure pressure of the cylinders. From this the ideal diametral interference, the best shrinkage radius can be obtained. Autofrettage has a negligible effect on increasing the pressure capacity of the compound cylinder, a compound cylinder with an appropriate shrinkage radius, as determined in this work, is superior to a similar single strengthened by autofrettage.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 04, April-2018

ACKNOWLEDGEMENTS

The author wish to thank Dr T.Christopher, the head of department of mechanical engineering, Government College of Engineering, Thirunelveli for his full support.

REFERENCES:

[1] G.H.Majzoobi, Experimental and finite element prediction of bursting pressure in compound cylinder. International Journal of Pressure Vessels and Piping 81 (2004) 889-896.
[2] G.H.Majzoobi, Optimisation of compound pressure cylinder, Journal of Achievements in Materials and Manufacturing Engineering Volume 15,Issue 1-2, March-(2006).
[3] J.M.Kihih ,Stress characterization of autofrettaged thick-walled cylinder,International journal of mechanical Engineering.
[4] T.Asser Brabin,Investigation on failure behavior of unflawed steel cylindrical pressure vessels using FEA . Multidiscipline Modeling in Mat. And str.(2009)29-42
[5] A.P.Beena Destructive tests of 15CDV6 steel rocket motor cases and their application to lightweight designA.P.Beena, International Journal of Pressure Vessels and Piping 62 (1995) 313-320.