A PAPER ON DESIGN AND ANALYSIS OF PRESSURE VESSEL

P.Palanivelu [1], R.Siva Prasad [2],

[1] PG Scholar, Department of Mechanical Engineering,
Gojan School of Business and Technology, Redhills, Chennai, India.
p_velu79@yahoo.com
[2] Assistant professor, Department of Mechanical Engineering,
Gojan School of Business and Technology, Redhills, Chennai, India.
spsivaprasath50@gmail.com

ABSTRACT

Objective: This Project investigates to determine the stress distribution and failure location for a given pressure vessel using Finite Element Analysis (FEA). Comparing the FEA results with analytical results shows that the FEA software predicted the failure location very well for the symmetric shaped pressure vessel. By modifying the design and keeping the same material of the given pressure vessel, we could reduce the induced stress in the pressure vessel or we can increase the working pressure. The results are to be compared on the basis of Maximum distortion energy theory also known as Von Mises-Hencky theory i.e. Von Mises yield criterion. And the deviation between the analytical results and the software result, around the critical location to be checked for the stress discontinuity. Since it is an axis-symmetric pressure vessel, it is allowed to be modelled as a 2D axis-symmetric model and analysis is done.

Keywords: Pressure vessel, Theories of failure, Yield Criterion, Puncture disc, Ellipsoidal dome, FEA model, Boundary condition

INTRODUCTION

A Pressure vessel is a closed container designed to hold gas or liquid at high pressure substantially different from the ambient pressure. They may be of any shape and range from beverage bottles to the sophisticated ones encountering in engineering construction. Many commonly used pressure vessels have well defined analytical equations that are used to determine their burst pressure and safety factors. Because these standard-shaped vessels have been studied for many years, their failure locations are also well documented.

STRESSES ON PRESSURE VESSEL

The stresses on vessels, produce changes in their dimensions known as strains. The determination of relationship between external forces applied on it and the stresses and strains within the vessel form the basis of this field of stress analysis. The basic interaction of stresses and strains is well illustrated by conventional tensile test specimen, from which we can pursue the stress analysis in the materials and study its nature based on the stress-strain curve.

THEORIES OF FAILURE

There are four important failure theories: maximum shear stress theory, maximum normal stress theory, maximum strain energy theory, and maximum distortion energy theory. Out of these four theories of failure, the maximum normal stress theory is only applicable for brittle materials, and the remaining three theories are applicable for ductile materials. Of the latter three, the distortion energy theory provides most accurate results in majority of the stress conditions.

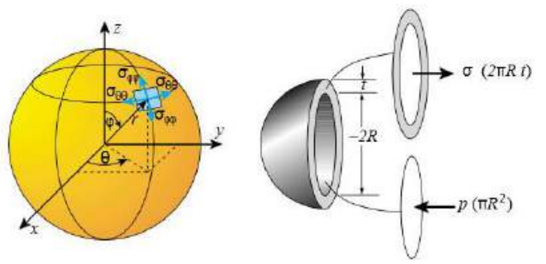
Maximum distortion energy theory - This theory is also known as shear energy theory or von Mises-Hencky theory. This theory postulates that failure will occur when the distortion energy per unit volume due to the applied stresses in a part equals the distortion energy per unit volume at the yield point in uni-axial testing. The total elastic energy due to strain can be divided into two parts: one part causes change in volume, and the other part causes change in shape. Distortion energy is the amount of energy that is needed to change the shape.

VON MISES YIELD CRITERION

In materials science and engineering the von Mises yield criterion can be also formulated in terms of the von Mises stress or equivalent tensile stress, , a scalar stress value that can be computed from the Cauchy stress tensor. In this case, a material is said to start yielding when its von Mises stress reaches a critical value known as the yield strength, . The von Mises stress is used to predict yielding of materials under any loading condition from results of simple uni-axial tensile tests. The von Mises stress satisfies the property that two stress states with equal distortion energy have equal von Mises stress.

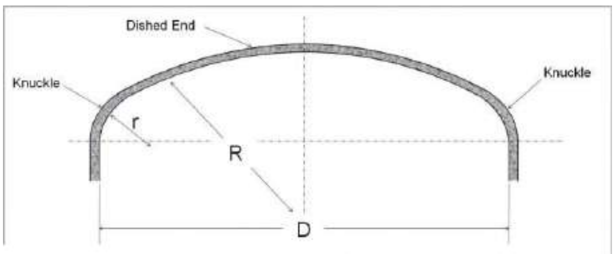
SPHERICAL VESSEL UNDER INTERNAL PRESSURE

When we take an plane element from the surface of the Spherical Vessel , it is subjected to biaxial stress namely hoop stress and longitudinal stress where both stresses are equal and constant over the entire vessel as shown in below figure.



σxx = σθθ = σL of sphere = σH of sphere = pa / h

ELLIPSOIDAL VESSEL UNDER INTERNAL PRESSURE



At crown , The biaxial stresses are:

$$\sigma_h = \sigma_L = \frac{pa^2}{2bh}$$

At equator,

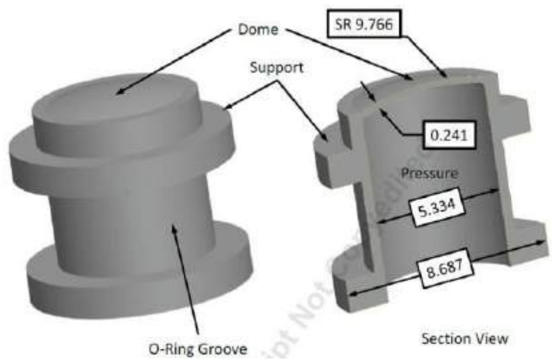
$$\sigma_h = \frac{pa}{h}\left(1 - \frac{a^2}{2b^2}\right)$$

$$\sigma_L = \frac{pa}{2h}$$

$$\tau_{max} = \frac{pa}{4h}\left(\frac{a^2}{b^2} - 1\right)$$

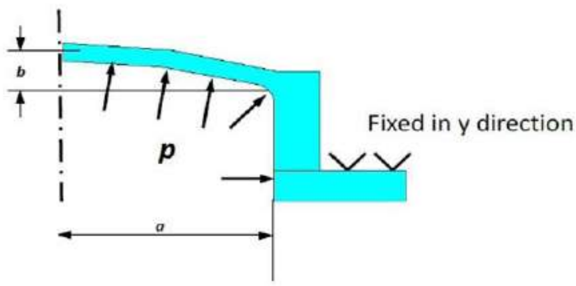
DESIGN APPROACH

PUNCTURE-DISC



Material	Titanium Commercially Pure (Grade 2)
Component	Puncture-disc
Density	4511.8 kg-m^{-3}
Modulus of Elasticity	106,868 Mpa
Poisson's Ratio	0.32
Yield strength	450 Mpa
Specification	ASTM B 348
Working pressure	1000Kpa

ELLIPSOIDAL DOME



From Figure,

a =5.334/2 = 2.667 mm

SR = 9.766 mm, b = $SR - \sqrt{SR^2 - a^2}$ =

$9.766 - \sqrt{9.766^2 - 2.667^2}$ = 0.371 mm

a/b ratio = 7.1886

Allowable stress = weld efficiency * yield strength = 0.78 * 450 = 350 MPa

MAJOR TO MINOR AXIS RATIO

As noted above, FSW can be a slower process In the set of equations the a/b ratio plays vital role in the dome of the pressure vessel. For extreme curvature the spherical segmented shapes can be considered as ellipsoidal segments for analytical relations.

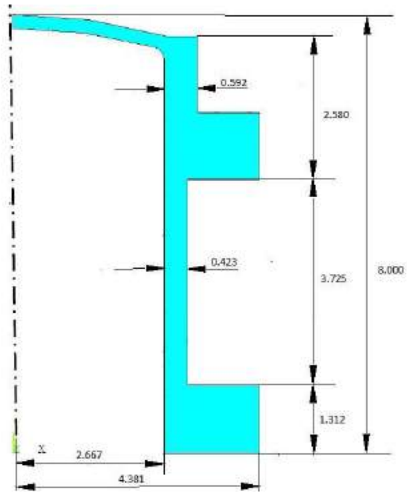
If we see those set of equations, the variation of a/b ratio affects those biaxial stresses. The hoop stress is tensile in crown but this decreases as equator is approached where it becomes compressive for a/b ratio greater than 1.42.

- Case 1: $\frac{a}{b} = 1$ The magnitude of biaxial stresses are equal and remains constant in all location.
- Case2: $\frac{a}{b} < 1.42$ maximum stress occurs in the crown location and biaxial stresses are always tensile in nature.
- Case3: $\frac{a}{b} > 1.42$ at this stage the hoop stress in the equator region becomes compressive and contributes to shear, and as the ratio increases, the location of maximum stress swifts from the crown to equator . Eventually buckle happens in the equator as maximum shear stress in equator is localised.

ALTERNATIVE DESIGN

In order to increase the working pressure or to reduce the induced stress for given working pressure, the design of the geometry of the pressure vessel can be changed by playing with parameters in the given equations. We may believe that thickness and a/b ratio helps us effectively to meet our expectations. But the reality is changing those parameters blindly we may get in trouble in stress localised in other locations.

For example, changing the thickness of the head blindly we get into trouble due to stress discontinuity in the junction of head and body. And more decrease in the a/b ratio causes stressed up in the lower bottom of the cylindrical vessel and we may be forced to add up thickness in the lower part as shown in figure.



DESIGN CALCULATION

EXISTING MODEL

Stresses in ellipsoidal head

At crown ,

The biaxial stresses are:

P = 1 MPa
a = 2.667 mm
b = 0.371 mm
h = 0.241 mm

$$\sigma_h = \sigma_L = \frac{pa^2}{2bh} = \frac{1*2.667^2}{2*0.371*0.241}$$

= 39.77 MPa

At equator, substituting the known values,

$$\sigma_2 = \sigma_h = \frac{pa}{h}\left(1 - \frac{a^2}{2b^2}\right) = -274.8 \text{ MPa}$$

(compressive)

$$\sigma_1 = \sigma_L = \frac{pa}{2h} = 5.5 \text{ MPa}$$

$$\tau_{max} = \frac{pa}{4h}\left(\frac{a^2}{b^2} - 1\right) = 140 \text{ MPa}$$

Von Mises equation is

σv2 = σ12 + σ22 - σ1σ2 = 38509.97

Von Mises stress σv = 196.239 MPa

Biaxial stresses in cylindrical portion
Lower portion where
h = 0.4233 mm
a = 2.667 mm
p = 1 MPa

σxx σL of cylinder = $\frac{pa}{2h}$ = 3.15 MPa

σθθ σH of cylinder = $\frac{pa}{h}$ = 6.3

Von Mises stress of cylindrical σv = 5.45 MPa
Radial growth in vessel Cylindrical portion:
E = 106,868 MPa
h = 0.4233 mm
μ = 0.32

δc = $\frac{pa^2}{2hE}(2-\mu)$ = 2.32 * 10^{-4} mm

ANALYTICAL RESULTS

- So the maximum induced stress is 196.239 MPa in the equator location of ellipsoidal head.
- Stress induced on crown location of head is 39.77 MPa.
- Radial growth in cylindrical portion is 2.32e-4 mm.
- We did take possible analytical results and we shall compare it with ANSYS results.
- It is safer as the induced stress is less than the allowable stress = 350MPa.

DESIGN MODIFICATION

We are not going to make changes in thickness of vessel for the sake of leaving the discontinuity issues untouched. So the a/b ratio is changed reasonably and to determine the induced stress for the given working pressure to compare with the existing model. For our new design a/b ratio is reduced to 3 so,
a = 2.667 mm, b = 2.667 / 3 = 0.889 mm
our new minor axis of ellipsoidal head is 0.889 mm

STRESSES IN MODIFIED ELLIPSOIDAL HEAD

At crown , the biaxial stresses are:

P = 1 MPa
a = 2.667 mm
b = 0.889 mm
h = 0.241 mm

σh = σL = $\frac{pa^2}{2bh} = \frac{1*2.667^2}{2*0.889*0.241}$ = 16.599 MPa

At equator, substituting the known values,

σ2 = σh = $\frac{pa}{h}\left(1-\frac{a^2}{2b^2}\right)$ = -38.73 MPa (compressive)

σ1 = σL = $\frac{pa}{2h}$ = 5.5 MPa

τmax = $\frac{pa}{4h}\left(\frac{a^2}{b^2}-1\right)$ = 22.13 MPa

von Mises equation is

σv2 = σ12 + σ22 - σ1σ2 = 1743.27

von Mises stress σv = 41.75 MPa

Radial growth in vessel

Cylindrical portion:

E = 106,868 MPa
H = 0.4233 mm
μ = 0.32
Substituting known values

Δc = $\frac{pa^2}{2hE}(2-\mu)$ = 2.32 * 10^{-4} mm

ANALYTICAL RESULT OF MODIFIED DESIGN

- So the maximum induced stress is 41.75 MPa in the equator location of ellipsoidal head.
- Stress induced on crown location of head is 16.599 MPa.
- Radial growth in cylindrical portion is 2.32 e^{-4} mm.
- We did take possible analytical results and we shall compare it with ANSYS results. It is safer as the induced stress is less than the allowable stress = 350 MPa.

OUTCOMES FROM ANALYTICAL RESULTS

- Radial growth is evaluated at the lower portion so that it gives results that are free from constraints, discontinuities.

- Though both the cases the working pressure is same but the stress induced in the latter model is lower than the former.
- Stress induced in modified model = 0.212 X stress induced in existing model, what we infer is from this is 4.7 times of working pressure of existing model can be given to new model i.e. 4.7 MPa.
- We compare based on von-mises stress criterion as it is very accurate and suitable for ductile materials.

FINITE ELEMENT ANALYSIS

FEA MODEL

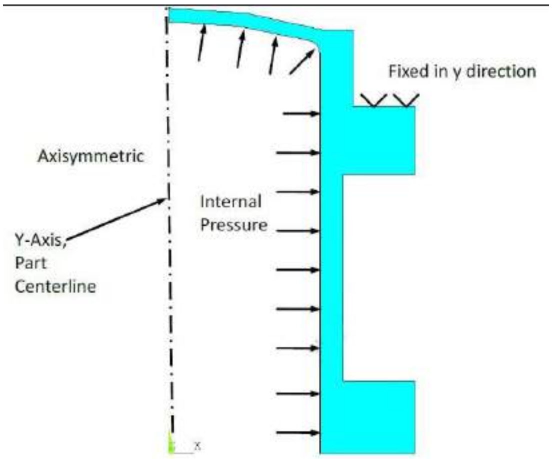
FEA model of existing pressure vessel completed using ANSYS 14.5 Mechanical software. The symmetrical shape of the puncture disc allowed it to be modelled as a 2-D axis symmetric model. The 2D puncture disc model utilized the ANSYS Plane 183 element. This element type is an 8-node quadratic element that can be used in an axis symmetric model. The mesh was highly refined in the area where the disc was expected to fail; the elements in the predicted failure location had a nominal edge distance of 0.0127mm.

MATERIAL MODEL

The linear elastic material properties for each material are listed in Table. The puncture disc is made from Grade-2 commercially-pure titanium in accordance with American Society for Testing and Materials ASTM B 348, both materials were treated as isotropic.

BOUNDARY CONDITIONS

The puncture disc has a surface pressure applied to the entire inside surfaces, as shown in Fig, A displacement constraint fixes the y-axis movement on the surface where it is held in the pressure vessel assembly. This surface is still allowed to move in the x-axis direction. There is also an axis symmetric constraint along the y-axis at the edge of the domed section. The axis is shown by the centre-line in Fig. Since this is an axis symmetric model, the left edge shown in Fig is the centre of the part. The axis symmetric boundary condition eliminates the need for any additional constraints to keep the model from moving in space.



MODELLING AND ANALYSIS PROCEDURE

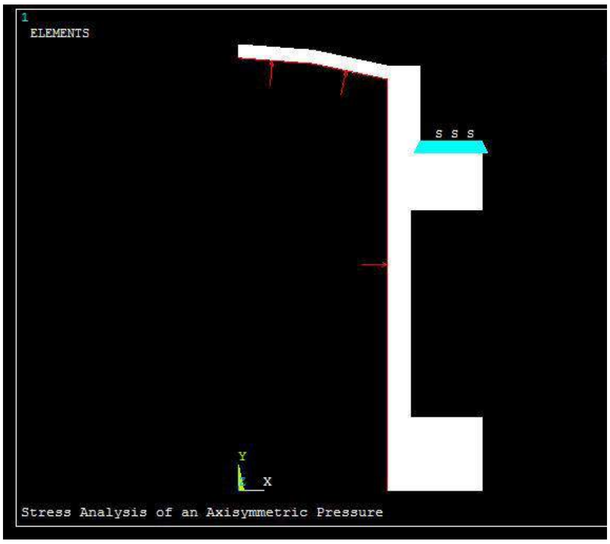
Since the models are axis symmetric, so the problem is considered as 2D problem. Modelling and analysis is carried out in Ansys APDL. Here the modelling is done by key points, these key points are determined from the given dimensions and making a frame of reference. One assumption is done that the curvatures of the pressure vessel head are considered as number of individual lines joining the key points and these key points satisfy the equation of circle and elliptical respectively.

Meshing is finely done to determine the failure location and maximum induced stress, it given 0.021 mm of fine mesh to make an accurate set of results.

EXISTING PUNCTURE DISC PRESSURE VESSEL

Key points	X coordinate e-3	Y coordinate e-3
1	2.667	
2	4.381	
3	4.381	1.31223
4	3.0903	1.31223
5	3.0903	5.0372
6	4.381	5.0372
7	4.381	6.2647
8	3.259	6.2647
9	3.259	7.61926
10	2.667	7.61926
11	1.3335	7.8992
12	0	8
13	0	7.759
14	1.3335	7.6582
15	2.667	7.3826

- These key points are entered ,
- Create lines through key points
- Create areas through lines
- MESHING the area of quad elements of size $0.012e^{-3}$ m
- Apply the constraints as displacement symmetry on line, select the support.
- Apply pressure value as 1000000 Pa on selecting inner lines



MODIFIED PUNCTURE DISC

PRESSURE VESSEL

- All the above procedures are same, the key points are given differently to model.

Key points	X coordinate e-3	Y coordinate e-3
1	2.667	
2	4.381	
3	4.381	1.31223
4	3.0903	1.31223
5	3.0903	5.0372
6	4.381	5.0372
7	4.381	6.2647
8	3.259	6.2647
9	3.259	7.61926
10	2.667	7.61926
11	2	8.20726
12	1.3335	8.38916
13	0.66675	8.479
14	0	8.50826
15	0	8.267
16	0.66675	8.23896
17	1.3335	8.14815
18	2	7.96626
19	2.667	7.37826

CONCLUSION

Based on the above design and analysis, the critical location in the pressure vessel lies along the equator of the pressure head and from where the crack propagation starts. The critical location in the pressure vessel lies along the equator of the pressure head and from where the crack propagation starts. Changing the thickness in design may produce discontinuity stress in the junction, thus by altering the a/b ratio reasonably we can get positive results. When a/b ratio is increased, we observe that lower part of the cylindrical portion gets stressed up and yet not significantly. Making such design, we can increase the working pressure without changing the material.

REFERENCES

[1] Failure Prediction of Pressure Vessels using Finite-Element Analysis -Christopher J. Evans, P.E. Dr. Timothy F. Miller, P.E.

[2] ASME Section III Treatment of Stress Distribution in Cylindrical Vessels With Symmetric Thin-Walled Discontinuity L.Ike Ezekoye, Gerald A.Riegel, David Ristau and Richard Way

[3] Theory and design of pressure vessels – John F Harvey P.E

[4] C.D.Miller,"Summary of buckling Tests on Fabricated Steel Cylindrical shells in USA"