Paper ID: MS-38

Stress Analysis of an Internal Grooved Pressure Vessel

Abdul Mohaimen Safi
Department of Mechanical Engineering, BUET, Dhaka- 1000, Bangladesh
E-mail: mohaimensafi@gmail.com

Abstract

Pressure vessels are used in a variety of application in both industry and private sector. They appear in these sectors as industrial compressed air receiver and domestic hot water storage tank. Until recently primary analysis method for calculating stress had been hand calculation and empirical curves. New computer advances have made numerical computation a powerful tool in the study of pressure vessel especially in determining stresses in grooves which are difficult to analyze by hand calculation. This paper presents a numerical investigation for stress distribution in a cylindrical pressure vessel having internal groove under the application of internal pressure. Solutions especially at the groove region which are observed as the critical zone are determined by finite element numerical technique. The effect of groove radius and the thickness ratio of the of the pressure vessel have also been studied.
.

Keywords: *Pressure vessels, internal groove, finite element technique.*

1. Introduction

Pressure vessels are closed containers designed to hold gases or liquids at a pressure different from the atmospheric pressure. They may be of any shape and size ranges from cold drink bottles to high pressure steam boilers used for engineering application. The word "Design" means not only to create, and calculate the detail dimensions of the pressure vessel but also to find out the mode of failure and selection of materials type and its environmental behavior [1]. The increasing industrial demand for pressure vessels which have application in chemical, nuclear, petrochemical, oil, power plant and military equipment have concentrated the attention of engineers on this particular area of engineering to find out most likely method of damage, method of stress analysis and reliability of results [2]. Though experimental methods give the most reliable results, it is very costly, as it requires special equipments, testing facilities etc. Analytical solution of every problem is almost impossible because of complex boundary conditions and shapes. For this reason the numerical methods had become the ultimate choice by the researchers in the last few decades. Invention and rapid improvement of the computing machines, i.e. sophisticated high performance computers, also played an important role for the increasing popularity of the numerical methods. Stress analysis of a pressure vessel having internal groove requires the solution of partial differential equations. There are various numerical methods available for the solution of partial differential equations. Among them most popular methods are: Finite Element Method (FEM) and Finite Difference Method (FDM). The finite element method is a numerical technique for obtaining approximate solution to a wide variety of engineering problems. Although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics because of its diversity and flexibility as an analysis tool. The finite difference model of a problem gives a point wise approximation to the governing equations. With finite difference techniques we can treat some fairly difficult problems; but for example, when we encounter irregular geometries or an unusual specification of boundary conditions, we find that finite difference techniques become hard to use. A finite element model of a problem gives a piece wise approximation to the governing equations. Since these elements can be put together in a variety of ways, they can be used to represent exceedingly complex shapes. In the present paper numerical simulation for a non-circular pressure vessel having internal groove under the application of internal pressure has been investigated using ANSYS one of the most popular finite element method software.

s

2. Equations for non-circular pressure vessel

For non-circular vessels, the hoop stress in an obround is comprised of a shape-dependent bending component superimposed on the pressure-induced membrane component. A representative obround shape is represented in figure 1 & the corresponding stress components at the noted critical points are as follows [3]

Membrane Stress (σ_m):

Locations A, B: $$(\sigma_m)_{A,B} = \frac{PR}{t} \quad (1)$$

Locations C, D: $$(\sigma_m)_{C,D} = \frac{P(R+L_2)}{t} \quad (2)$$

Bending Stress (σ_b):

Locations A, B: $$(\sigma_b)_{A,B} = \pm \frac{PL_2RC_1c}{6AI} \quad (3)$$

Locations C, D: $$(\sigma_b)_{C,D} = \pm \frac{PL_2c}{6I}\left[3(L_2+2R) - \frac{C_1}{A}\right] \quad (4)$$

Where,

$$I = \frac{t^3}{12}$$

$$C_1 = L_2^2(2+3\pi) + 12R^2$$

$$A = 2\left(\frac{L_2^2}{R}\right) + \pi R$$

$C = \frac{t}{2}$ =distance from neutral axis to outer surface

Total Hoop Stress (σ_t):

Locations A, B: $$(\sigma_t)_{A,B} = (\sigma_m)_{A,B} + (\sigma_b)_{A,B} \quad (5)$$

Locations C, D: $$(\sigma_t)_{C,D} = (\sigma_m)_{C,D} + (\sigma_b)_{C,D} \quad (6)$$

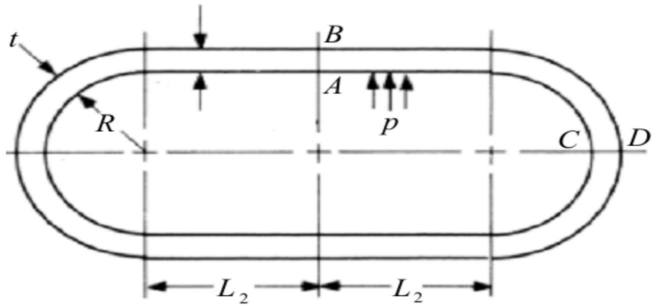

Fig. 1. Non-circular pressure vessel

3. Model development

The geometry of the problem is shown in Fig 2. As the vessel is axially symmetric about its central axis, an axisymmetric analysis was performed using two-dimensional, 8-node quadrilateral elements (Plane 82) with the axisymmetric option activated. In addition, the vessel is symmetric about a plane through the center of the cylinder. Thus, only a quarter portion of the vessel was modeled as indicated by the shaded area in Fig 2. Once the axisymmetric option was invoked, ANSYS automatically applied axisymmetric boundary conditions along the Y-axis. The material of the plate is high strength alloy steel; Poisson's ratio ν=0.33, Young's modulus E=200 GPa. Uniform Pressure was applied on all lines that make up the inner surface of the vessel.

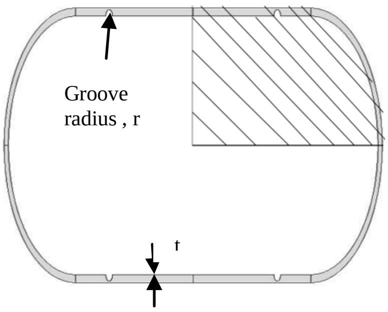

Fig. 2. Model Geometry

4. Meshing

Created geometrical model is discretized into nodes and elements. The process is called meshing. The Ansys program can automatically generate nodes and elements, provided that element size is specified. The element size controls the fineness of the mesh. The smaller the element size the finer the mesh. For the present problem 7200 elements have created, density of mesh at the groove is shown in Fig 3.

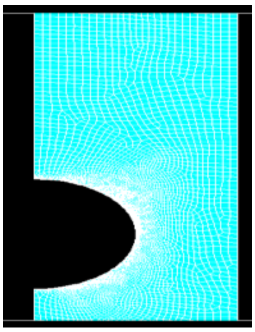

Fig. 3. Density of mesh at the internal groove

PLANE 82 (8-nod 2-D) elements shown in Fig 5 have been used in the analysis. PLANE 82 elements provide more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries. The element may be used as a plane element or as an axis symmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. [4]

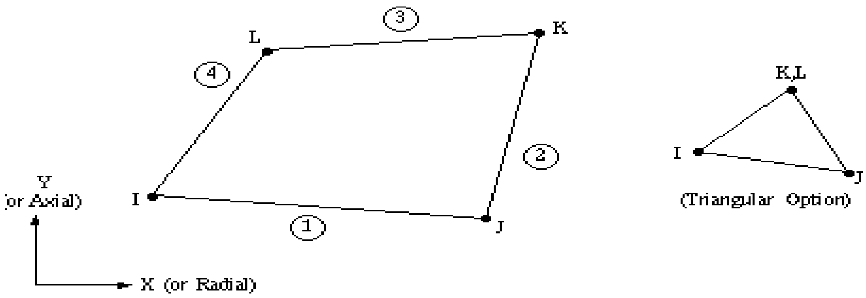

Fig. 4. PLANE 82 elements with 8 nodes

5. Results and discussions

The solution of stress components σ_r, σ_l and σ_t for each nodal point within the elastic limit has been obtained. The variations of values of σ_r, σ_l and σ_t with the change of non-dimensional ratio x/t were plotted for some selected sections. The selected sections are shown in Fig. 6.

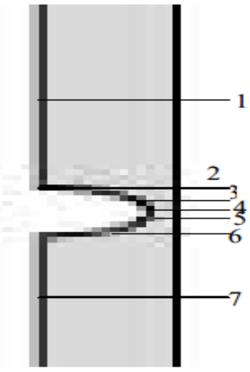

Fig. 5. Selected sections for studying the distribution of stresses

The distribution of stress component σ_r (radial stress) for some selected sections are shown in Fig. 7. For a constant value of x/t the maximum value of σ_r is obtained at the midpoint along the groove. The value of σ_r is zero outer most ends which satisfy the applied boundary condition.

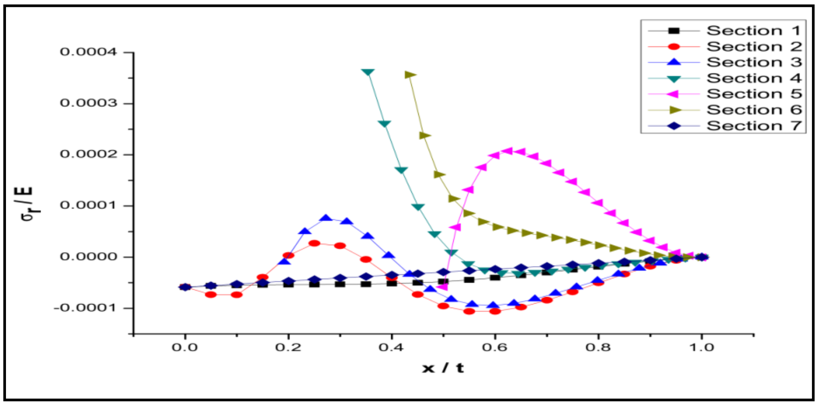

Fig. 6. Distribution of σ_r

The distribution of stress component σ_l (longitudinal stress) for some selected sections are shown in Fig. 8. For a constant value of x/t the maximum value of σ_l is obtained at the midpoint along the groove. It is found that for section 1 and 7 the value of σ_l is almost identical. So the distribution of stresses varies around the groove and becomes uniform large distance away from the groove.

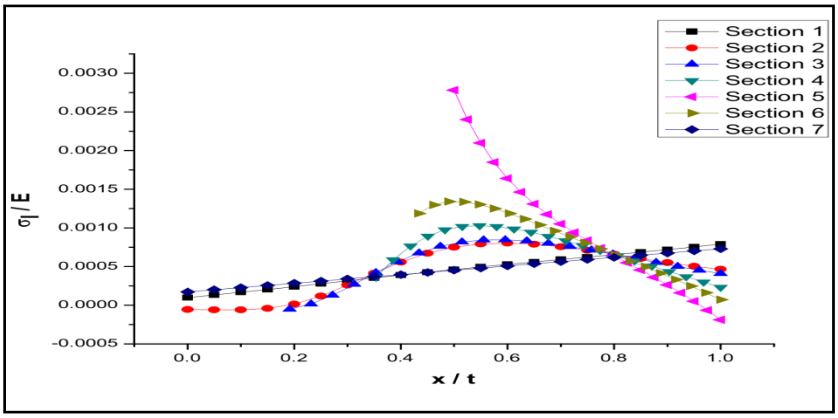

Fig. 7. Distribution of σ_l

The distribution of stress component σ_t (hoop stress) for some selected sections are shown in Fig. 9. For a constant value of x/t the maximum value of σ_t is obtained at the midpoint along the groove. It is observed that only section 1 and 7 the stress distribution is linear which indicate that distribution of stresses varies around the groove.

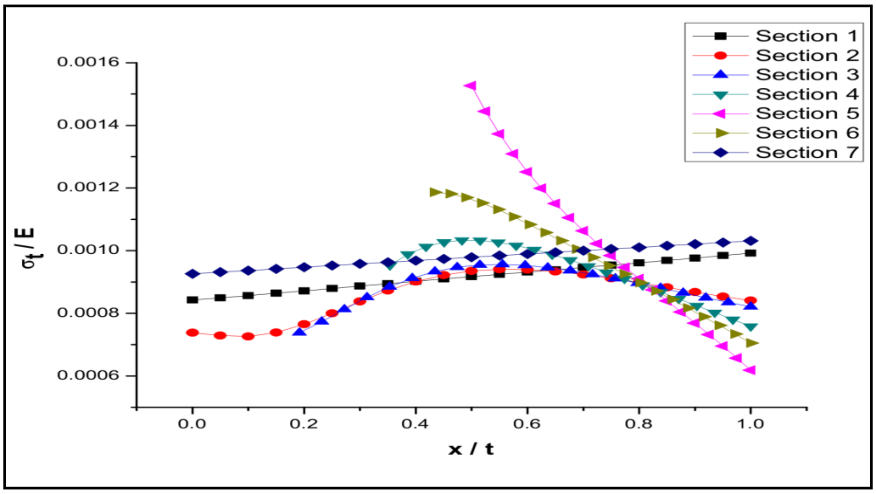

Fig. 8. Distribution of σ_t

The effect of degree of groove radius to thickness ratio on the pressure vessel

The effect of degree of groove radius to thickness ratio is studied by plotting the variations of values of σ_r, σ_l and σ_t with the change of non-dimensional ratio r/t for section 5. The distribution of stress component σ_r (radial stress) with the change of r/t is shown in Fig. 9. As appears for constant x/t the value of σ_r increases as the value of r/t increases for the same loading. So for the internal grooved pressure vessel r/t is an important factor.

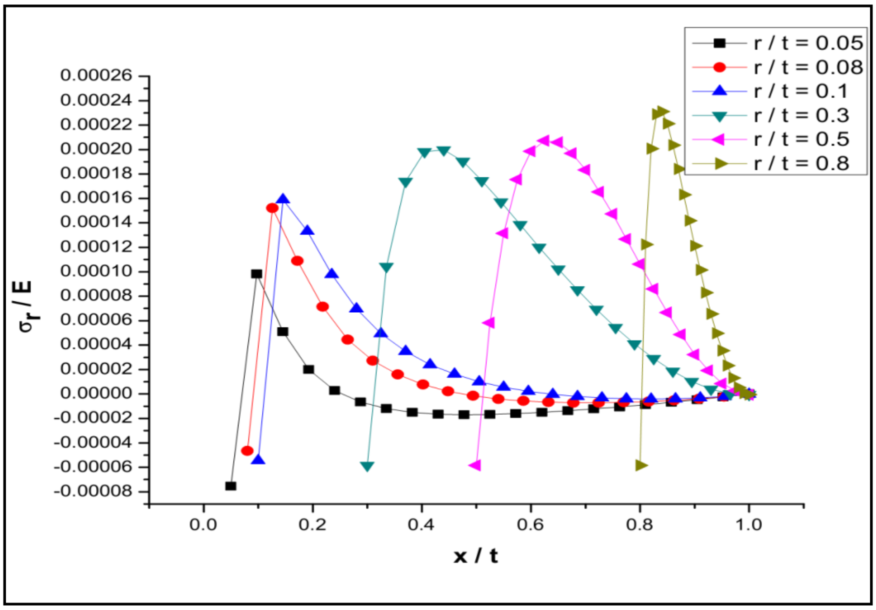

Fig. 9. Distribution of σ_r with the change of r/t

The distribution of stress component σ_l (longitudinal stress) with the change of r/t is shown in Fig. 10. It is observed that for constant value of x/t value of σ_l increases with the increase of r/t. It is found that with the decrease r/t for the same loading the stress distribution becomes identical as in case for r/t = 0.05, r/t = 0.08 and r/t =0.1. It is also appears that with increase of r/t curves becomes parallel to Y-Axis.

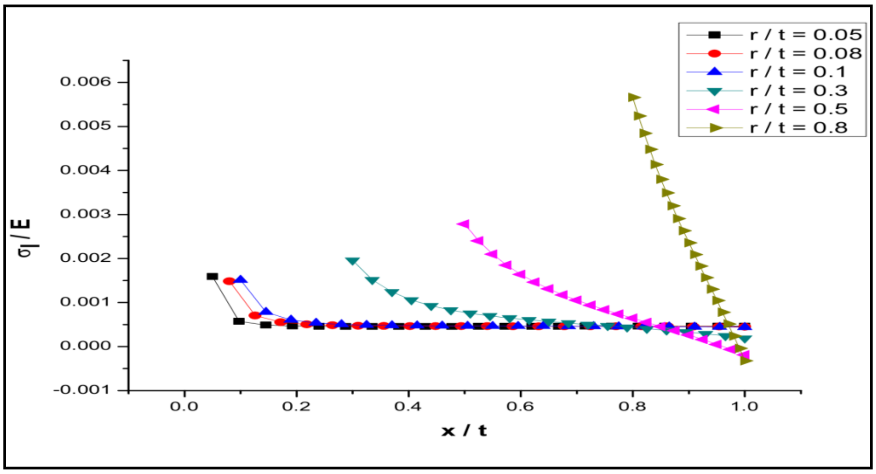

Fig. 10. Distribution of σ_l with the change of r/t

The distribution of stress component σ_t (longitudinal stress) with the change of r/t is shown in Fig. 11. It is observed that for constant value of x/t value of σ_t increases with the increase of r/t. This graph nature is similar to the graph plotted in Fig. 10. So from both graph it can be said that r/t ratio should be less than 0.3 for designing pressure vessel having internal groove.

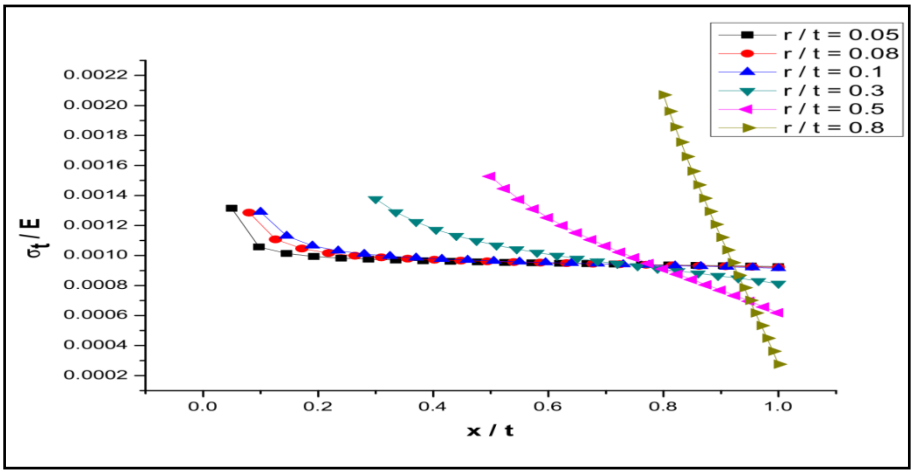

Fig. 11. Distribution of σ_t with the change of r/t

6. Conclusions

The present study provides the solution for stress components of a non-circular pressure vessel having internal groove subjected to an internal pressure using finite element approach. Both quantitative and qualitative results of pressure vessel obtained which establish the soundness and appropriateness of the present finite element approach.

7. References

[1] Vikram V. Mane, Vinayak H.Khatawate, Ashok Patole, "Finite Element Analysis of Ellipsoidal Head Pressure Vessel", Interntional Journal of Engineering and Research Applications (IJERA) ISSN: 2248-9622, 2012.

[2] Donatello Annaratone, Pressure Vessel Design, Springer, 3rd edition, pp. 2-5, 2007.

[3] Jonathan C. Wang, "Stress Analysis of an Elliptical Pressure Vessel Under Internal Pressure", Rensselaer Polytechnic Institute, 2005.

[4] "ANSYS" Theory Manual and Expanded ANSYS Workbook