ISSN : 2248-9622, Vol. 6, Issue 8, (Part -4 August 2016, pp.32-38

RESEARCH ARTICLE OPEN ACCESS

Stresses Analysis of Petroleum Pipe Finite Element under Internal Pressure

Dr.Ragbe.M.Abdusslam
Eng. Khaled.S.Bagar

ABSTRACT
This paper described the results of a nonlinear static mode within ANSYS of elastic and elastic-plastic behaviour of thin petroleum pipe that is subjected to an internal pressure and therefore a linear stress analysis performed using ANSYS 9.0 finite element software Such an analysis is important because the shape of most structures under internal pressure is cylindrical[1].
In this paper is considered only. Elastic and elastic-plastic finite element analysis is used to predict the principle stresses, effective stress results are compared with those obtained from theatrical equations in order to predict the limit and failure loads for this type of loading also the relationships between redial, hoop stresses and displacement has been used to develop a through understanding. The analysis was completed using ANAYS Version 9.0. (a finite element program for Microsoft Windows NT). The program allows pre-processing, analysis and post-processing stages to be completed within a single application. The program can be used to model a large number of situations including buckling, plastic deformation, forming and stress analysis problems.
$r_o = 609.6\,mm$ objected to an)In this study ,a thin pipe of internal radiu $r_i = 596.9\,mm$ and of external internal pressure $P_i = 4.83\ N/mm^2$

which is gradually increased to near the ultimate load that may be sustained by the pipe. The pipe is modelled as an elasto-plastic material using the Von Mises yield criterion which is normally used for metallic materials[2]. The specification of the load in several increments enables the spread of the plasticity to occur gradually and its effect on the stress distribution to be assessed. Key words: finite element analysis, elastic-plastic behavior, thin walled pipe equivalent stress, TWT.

I. Introduction

The most structures under internal pressure take cylindrical shape like as the boilers in thermal power plant, aerosol cans and gas cylinders, in this study one of the kind of this shape[3].
Elastic and elastic-plastic finite element analysis is used to predict the equivalent and principle stresses and results are compared with those obtained from theatrical equations. Theresearcher most understand mechanics properties of materials and stress-strain diagram and the principle stresses and strains because all the design of mechanical machines must have high strength to support external loads before it collapse.

II. Thin- walled pipe

A thin--walled pipe is one where the thickness of the wall is smaller than one-tenth of the radius[3] . In this study a thin petrolem pipe made from steel conveuor for oil extends from waha field to port of sidra as shown thickness 6.65mm, (in fig 1 of internal radius $r_i = 596.9\,mm$, external radius , $r_o = 609.6\,mm$
$L = 269000\ mm$ subjected to an internal pressure (4. 83N/mm²) as shown)
In fig 2.

Fig 1. petroleum pipe

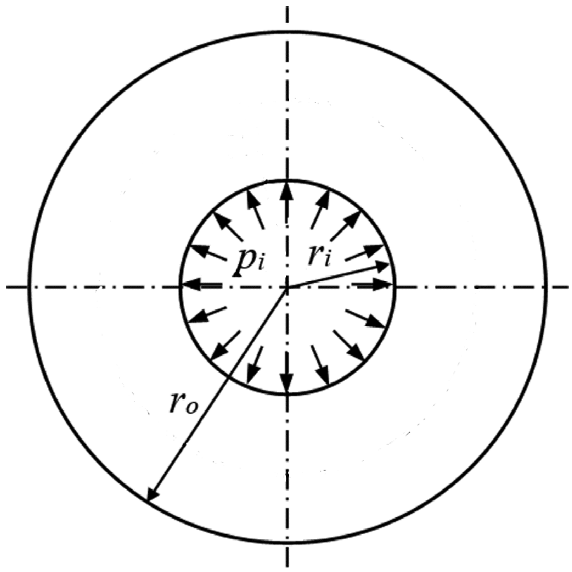


Fig 2 : thin walled pipe

2.1 Stresses in thin walled pipe

A thin walled pipe subject to an internal pressure has three principal stresses will be Hoop stress, axial stress and Longitudinal stress [3],[4] as shown in fig3. The stress conditions occur throughout the section and vary primarily relative to the radius.

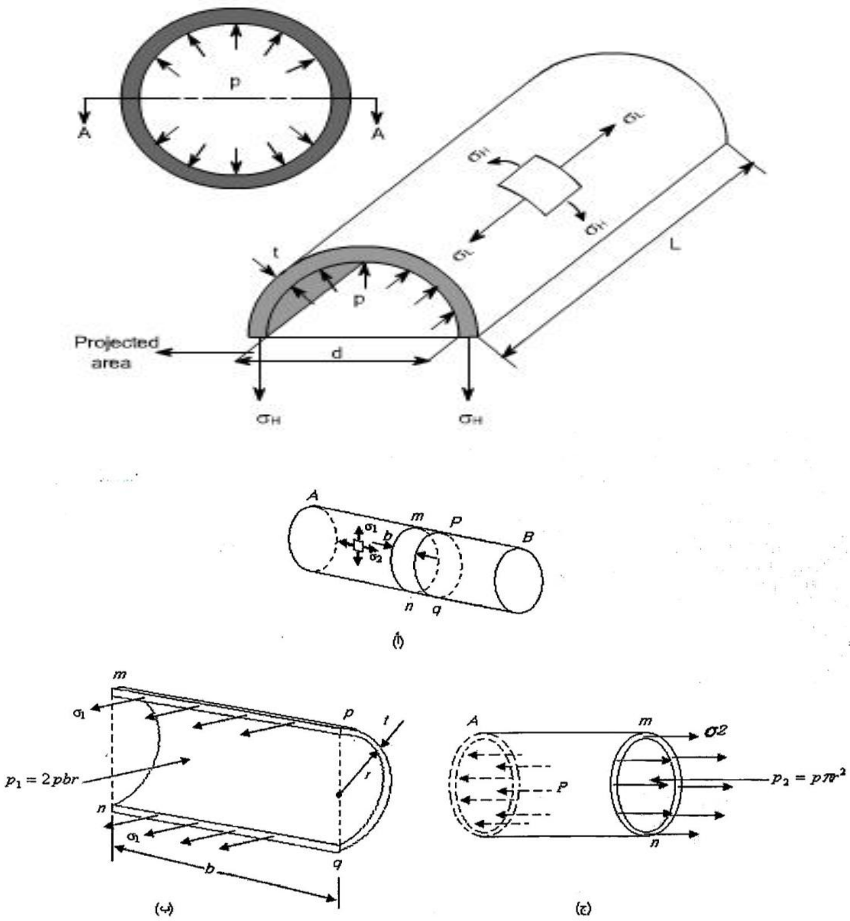


Figure 3 stresses in thin6t\ walled pipe

2.1.1 Hoop Stress

The hoop stress can be expressed as.

$\sigma_h = p\, d / 2\, t$ *(1)*

where

σ_h = hoop stress (MPa, psi) = σ_1

p = internal pressure in the tube or cylinder (MPa, psi)
d = internal diameter of tube or cylinder (mm, in)
t = tube or cylinder wall thickness (mm, in

$$\sigma_1 = \frac{P D_i}{2t} = \frac{4.83 \times 596.9}{2 \times 6.35} = 227.01\ \mathrm{N} / mm^2$$

2.1.2 Longitudinal stresses

The longitudinal stress can be expressed as.

$\sigma_l = p\, d / 4\, t$ *(2)*

where

σ_l = longitudinal stress = σ_2 *(MPa, psi)*

2.2.3 Axial stress

The axial stress can be expressed as:

$$\sigma_3 = -\frac{P}{2} \qquad 3)$$

$$\sigma_3 = -\frac{P}{2} = -\frac{4.83}{2} = -2.415\ \mathrm{N} / mm^2$$

Where

σ_3 Axial stress (MPa))= σ_t

The equivalent stress can be expressed as.

$$\sigma_{eq} = \frac{1}{\sqrt{2}} \times \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} \qquad \text{--- (4)}$$

$$\sigma_{eq} = \frac{1}{\sqrt{2}} \times \sqrt{(227.01 - 113.5)^2 + (113.5 + 2.415)^2 + (-2.415 - 227.01)^2} = 198.691\ \mathrm{N} / mm^2$$

The stresses on the pipe can be seen in table 1

Table 1the principle stresses and equivalent stress

	σ_1 N/mm²	σ_2 N/mm²	σ_3 N/mm²	σ_{eq} N/mm²
m	227.01	113.505	-2.415	198.691

2.2 Analytical solution

For a TWT with the following parameters the principal stresses calculated by the model equations are given in Table1 . The sketch of the tube half section is shown in fig 4

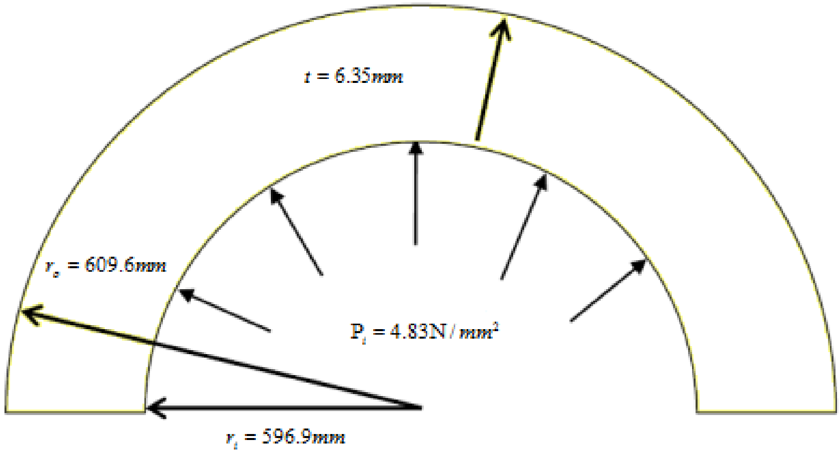


Fig..4 Half model of the pipe section subjected to internal pressure pi

2.2.1. Finite element modeling
The TWT as shown in Fig. 4 was numerically analyzed by finite element method and the results were compared with the analytical solution. The commercially available ANSYS 9.0 finite element software was used for this purpose. Two dimensional finite element analysis (FEA) was conducted using 4-noded quadrilateral elements under plane-strain condition[2]s.

2.2.1.1 Model Geometry
Fig. 5 shows the two dimensional model geometry of the tube used for FEA. The symmetry of the tube was taken advantage of and a solid model for a half section of th e tube was created in the ANYSYS pre-processor. The same symmetry conditions

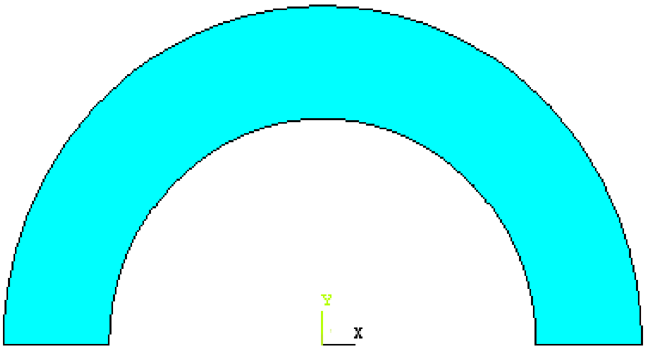


Fig 5 TWT Model used for FEA

2.2.1.2 Material properties
During FEA, an isotropic material with modulus of elasticity *E* = 71 GPa and Poisson's ratio, □ = 0.33 was used [5]

. ***2.2.1.3 Element selection and meshing***
The TWT was meshed using two dimensional 4-noded, PLANE42 solid elements. The parametric study was conducted to see the effects of element size on the results. Meshed model is shown in Fig. 6.

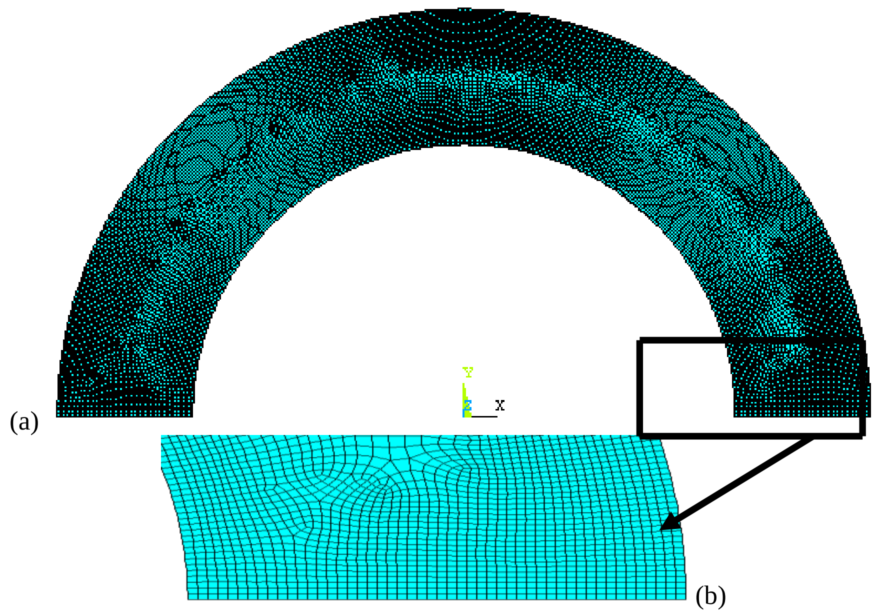


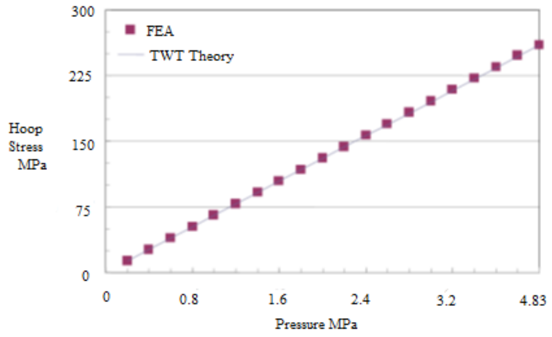
Fig.6. a) Meshed model using PLANE42 element b) magnified view of boxed area; element size is 0.5 mm

The von Mises stress distribution obtained after solution is shown in Fig. 7

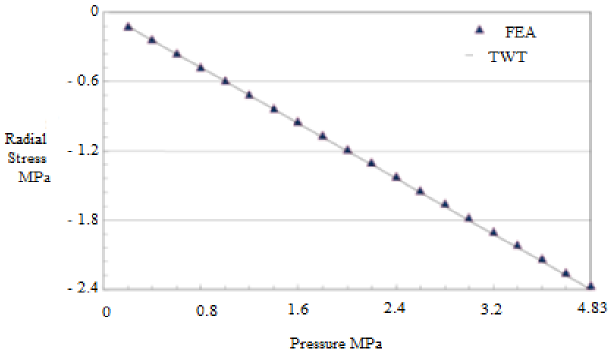
Fig.7. Static loading – Nodal solution showing von Mises stress distribution at internal pressure of a) 4.83MPa

III. Comparison of the analytical and numerical results

The results of the stress distribution obtained from analytical (thin-walled pipe theory, and numerical techniques were compared to see the validity of the model. Figs. 8a and 8b show the graphical presentation of the analytical and the FEA results of stress versus internal pressure at inner radius. The stress variation along the wall thickness of the tube obtained from the two methods is shown in Fig. 9.

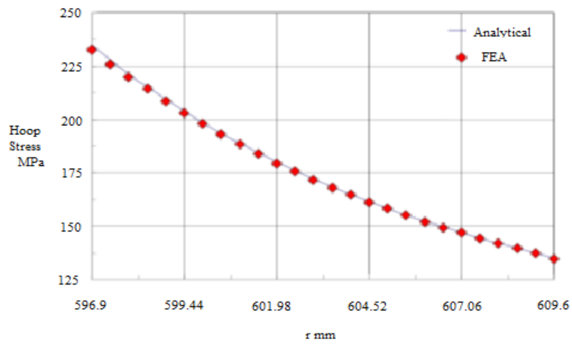


(a)

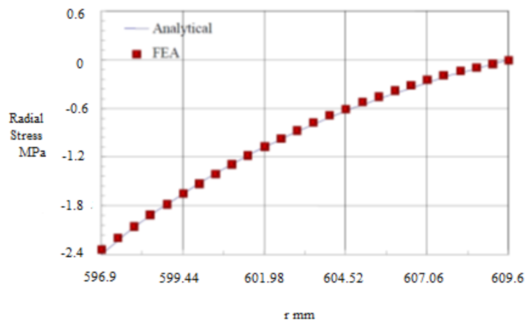


(b)

Fig. 8. Stress versus internal pressure - comparison of the two results at inner radius a) hoop b) radial



(a)



(b)

Fig..9 Stress variation along the wall thickness of the pipe obtained from the two methods at an internal pressure of 4.83 MPa a) hoop b) radial

In Fig. 8, both the hoop and radial stresses obtained from the analytical and FEA methods change linearly with the applied internal pressure. It can be seen that the results obtained from the two techniques are in good agreement. Fig. 9a shows a gradual decrease in the hoop stress from inner to outer radius. The highest tangential (hoop) stress is found at the inner radius i.e. at the inner wall of the pipe. In Fig. 9b the change in radial stress along the wall thickness of the hoop is presented. A compressive stress is found which varies from 4.83 MPa at the inner radius to a value of 0 MPa at the outer radius. Again the results obtained from the two techniques and presented in Figs. 9a and 9b are in fairly good agreement. This concludes that the half model used for the stress analysis is providing satisfactory results and can be used for the analysis of the pipe with internal crack.

REFERENCES

[1]. ANSYS User Manual Version 9.0, (a finite element program for Microsoft Windows NT).
[2]. Zienkiewicz, O. C. 1984The finite element method, McGraw -Hill
[3]. Black, P.H and Adams, E 1970 Machine design
[4]. Karwa, R 2000 Machine Design
[5]. James , M,Gere . 2004 . Mechanics of Materials . Sixth edition . Thomson Learning Academic . Resource Center . USA.