

F.E.M. ANALYSIS OF THE FRACTURE CONDITIONS AT THIN WALL ALUMINIUM PIPES

VALENTIN ZICHIL
VASILE NASTASESCU
GHEORGHE PINTILIE

CARMEN SAVIN
ADRIAN JUDELE
AURELIAN ALBUT

*University of Bacau, Engineering Faculty
600115 – Bacau, Marasesti no. 157, Romania*

Abstract

The paper presents the installation and the methodology for numerical analysis, using finite elements method that has been used for the breaking test of the aluminum pipes and also the general characteristics of the experimental results. The tested pipes have the same diameter $D = 18$ mm, but different wall-thickness: 2 mm for the civil construction pipes and 0.9 mm for the hydro melioration pipes. The time - pressure diagram is presented. The value of the breaking pressure for each tested pipe is determined, first from the theoretical approach and second, from the experiment. The solutions based on the classic beam structural theory and the detailed finite element analysis results have been confronted.

Keywords: Thin wall pipe, pressure elongation, beam theory, finite element analysis, commercial software package, welded steel pipes, hydrostatic breaking test.

1. INTRODUCTION

The paper presents a numerical study regarding to the behavior of a straight tube and an elbow under internal pressure. Both of them were modeled by finite elements of different types: SHELL and SOLID. The study started with a static analysis, the material being considered an elastic and isotropic one. Then, special material models were used, dynamic loading was considered. In the same conditions, an analysis of geometric parameters influence was performed [1]. Different ratio of the curvature radius and internal diameter (R/D) were considered and also different ratio of the curvature radius and the thickness (R/T) were considered too. In all the cases the results were about the same but the computer time for solving the problem was different, the shell elements being better from this point of view [2].

2. LOAD CONDITIONS

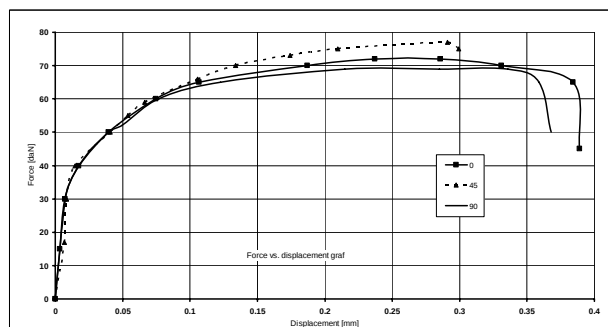


Fig. 1. The characteristic curve

During the test made by the authors, the material used for both type of pipes was aluminum. The characteristic curve of the material is shown in figure 1, based on the results presented in table 1. Those tests were made in three pressure load conditions: the first without pressure, the second at the pressure of 20 MPa and the third at the pressure of 40 MPa. A cylindrical (thin wall) shell subjected to uniform internal pressure is characterized by a plane stress state (figure 2, a); the longitudinal/axial (σ_x) and circumferential/hoop (σ_h) normal pressure

stresses are generated within the cylindrical wall. The longitudinal/axial (ε_x) and circumferential/hoop (ε_h) linear elastic strains are developed accordingly. The corresponding static and physical equations may be arranged as follows [3]:

Table 1.

Elem	EPSX	EPSY	EPSZ	GMXY	GMXZ	GMYZ
1	-5.130e-007	-1.868e-007	9.435e-007	3.007e-008	-4.732e-010	-2.752e-008
101	-4.981e-007	-2.027e-007	9.130e-007	2.990e-008	-5.271e-010	-3.530e-010
201	-5.050e-007	-1.934e-007	9.089e-007	3.403e-008	-2.177e-010	-1.332e-009
301	-5.091e-007	-1.941e-007	9.099e-007	3.412e-008	1.992e-010	-1.097e-009
401	-5.105e-007	-1.935e-007	9.099e-007	3.375e-008	7.534e-011	-6.036e-010
501	-5.111e-007	-1.934e-007	9.099e-007	3.367e-008	-6.425e-011	2.805e-010
601	-5.112e-007	-1.936e-007	9.097e-007	3.425e-008	-3.813e-010	1.024e-009
701	-5.076e-007	-1.924e-007	9.076e-007	3.535e-008	-2.246e-010	6.510e-010
801	-4.935e-007	-2.016e-007	9.046e-007	3.310e-008	2.366e-010	-6.281e-009
901	-4.618e-007	-1.900e-007	9.014e-007	2.384e-008	4.376e-009	-1.343e-008

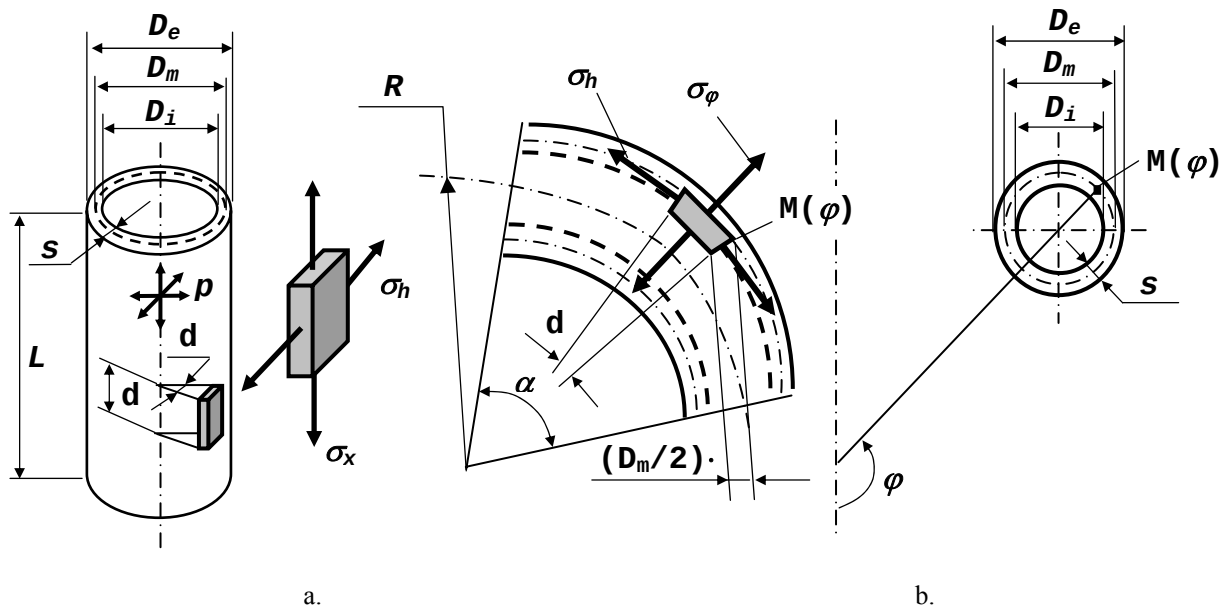


Fig. 2. Thin or medium wall tubular components subjected to uniform internal pressure load: a – cylindrical shell component (thin wall straight pipe); b – toroidal shell component (thin wall bend or elbow); D_e , D_i , D_m – outside, inside and medium tubular diameter; s – nominal wall thickness; L – cylindrical component length; R – bend/elbow curvature radius; (x, α) – cylindrical shell coordinates; (ϕ, α) – toroidal shell coordinates; $M(\phi)$ – current point belonging to bend/elbow component; p – internal gauge pressure; σ_x – longitudinal/axial normal pressure stress corresponding to cylindrical shell; σ_ϕ – meridian normal pressure stress corresponding to toroidal shell; σ_h – hoop normal pressure stress

$$\sigma_x = \frac{p \cdot D_m}{4 \cdot s} \quad ; \quad \sigma_h = 2 \cdot \sigma_x - \frac{p \cdot D_m}{2 \cdot s} \quad (1)$$

$$\varepsilon_x = \frac{\sigma_x - \mu \sigma_h}{E} = \frac{1 - 2 \cdot \mu}{2} \cdot \frac{\sigma_h}{E} \quad ; \quad \varepsilon_h = \frac{\sigma_h - \mu \sigma_x}{E} = \frac{2 - \mu}{2} \cdot \frac{\sigma_h}{E} \quad (2)$$

Similarly, a toroidal (thin wall) shell subjected to uniform internal pressure is characterized by a plane stress state too (figure 2, b); the meridian (σ_ϕ) and circumferential/hoop (σ_h) normal pressure stresses are generated

within the toroidal wall. The meridian (ε_φ) and circumferential/hoop (ε_h) linear elastic strains are developed accordingly. The corresponding static and physical equations may be arranged as follows [1]:

$$\sigma_\varphi(\varphi) = \frac{4 \cdot R + D_m \cdot \sin \varphi}{2 \cdot R + D_m \cdot \sin \varphi} \cdot \frac{p \cdot D_m}{4 \cdot s} \quad ; \quad \sigma_h = \frac{p \cdot D_m}{4 \cdot s} \quad (3)$$

$$\varepsilon_\varphi(\varphi) = \frac{\sigma_\varphi - \mu \sigma_h}{E} = \left[\frac{4 \cdot R + D_m \cdot \sin \varphi}{2 \cdot R + D_m \cdot \sin \varphi} - \mu \right] \cdot \frac{p \cdot D_m}{4 \cdot E \cdot s} \quad (4)$$

$$\varepsilon_h(\varphi) = \frac{\sigma_h - \mu \sigma_\varphi}{E} = \left[1 - \mu \cdot \frac{4 \cdot R + D_m \cdot \sin \varphi}{2 \cdot R + D_m \cdot \sin \varphi} \right] \cdot \frac{p \cdot D_m}{4 \cdot E \cdot s} \quad (5)$$

Within above formulas, E and μ represent Young modulus and Poisson ratio respectively.

3. SIMULATION AND RESULTS

For performing the dynamic and nonlinear analysis, a bilinear plastic kinematic material model, strain rate dependent, was used. This material type is one of the most used material models, adopted for dynamic and nonlinear analysis, just in the case of impact problems. The elastic plastic with kinematic hardening model was formulated by Krieg and Key and it is implemented in the most powerful software for structure dynamic nonlinear analysis. Using the elastic plastic with kinematic hardening material model a lot of other information can be obtained. For example, we can get the variation of the internal energy during the analysis time (figure 6) which praises the vibration phenomenon. A very important aspect in such analysis is to find out the pressure which could produce a fracture of the material. For this structure, for a ratio $R/D=1.0$ such an internal pressure has the value of 108 MPa. Figure 11 presents the elbow in the damage state.

Time = 4.0047e-005
Contours of Effective Stress (v-m)
max (pt. value)
min=1.39122, at elem# 21881
max=263.753, at elem# 21999

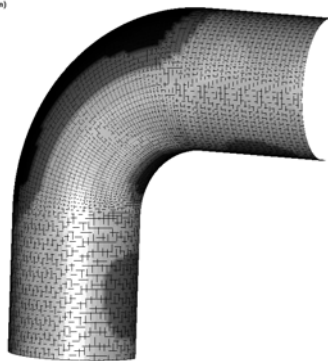
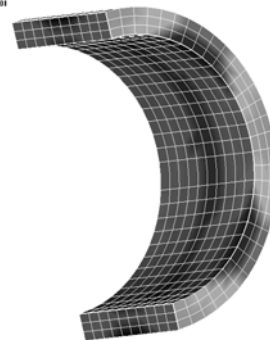


Fig. 3. Equivalent stress after von Mises criterion

Fringe Levels
2.638e+002
2.513e+002
2.388e+002
2.264e+002
2.139e+002
2.015e+002
1.890e+002
1.766e+002
1.641e+002
1.517e+002
1.392e+002

Time = 8.6722e-006
Contours of Effective Stress (v-m)
max (pt. value)
min=1.62937e+002, at elem# 536
max=3.37898e+006, at elem# 3108



Fringe Levels
3.379e+006
3.077e+006
2.776e+006
2.474e+006
2.172e+006
1.871e+006
1.569e+006
1.268e+006
9.659e+005
6.642e+005
3.626e+005

Fig. 4. Equivalent stress after von Mises criterion

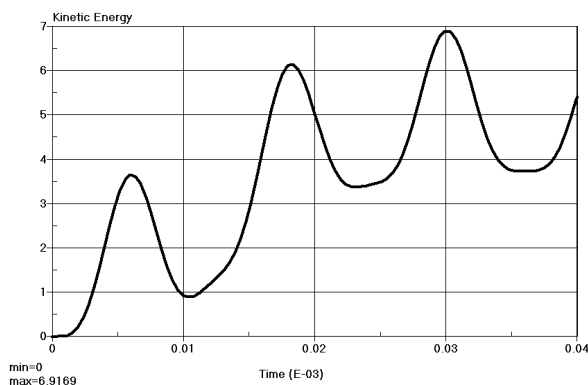


Fig. 5. Variation of the kinetic energy

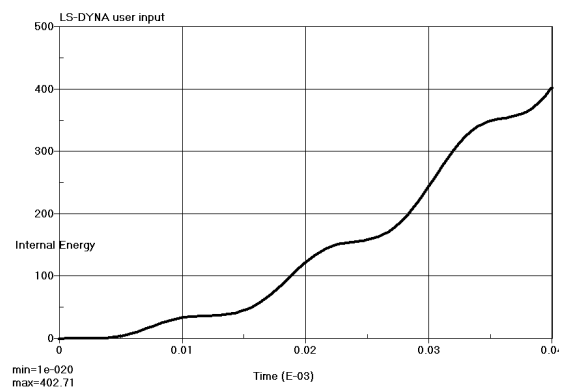
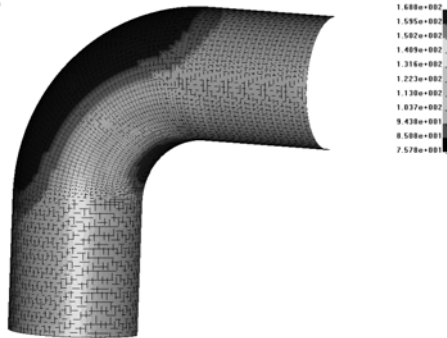


Fig. 6. Variation of the internal energy

Time = 4.8847e-005
Contours of Max Prin Dev Stress
min ip1 value
max=75.7772, at elem# 2823
max=168.791, at elem# 22398

Fig. 7. Stress on the 1st principal direction

Time = 4.8847e-005
Contours of XY-stress
min ip1 value
max=119.931, at elem# 15118

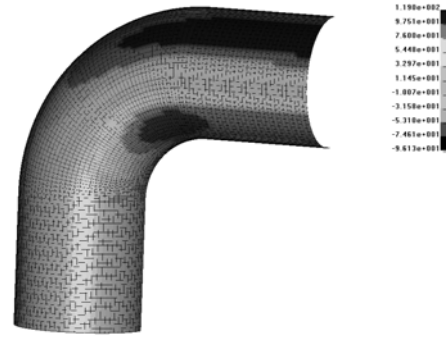
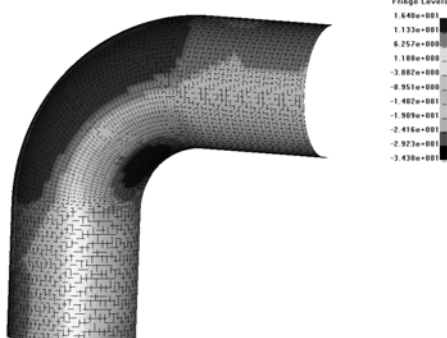


Fig. 8. xy stress

Time = 4.8847e-005
Contours of 2nd Prin Dev Stress
min ip1 value
max=14.2576, at elem# 22398
max=16.3855, at elem# 18346

Fig. 9. Stress on the 2nd principal direction

Time = 4.8847e-005
Contours of YZ-stress
min ip1 value
max=53.5488, at elem# 22408

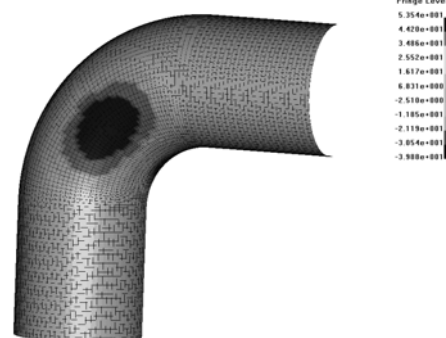


Fig. 10. yz stress

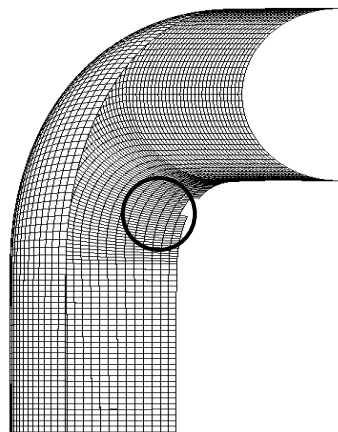


Fig. 11. The elbow damage state

4. CONCLUSIONS

The straight pipelines and elbows under internal pressure are a special structure often used in many industrial or private activities. As we can see the using only the linear behavior of the material leads us to an inefficient using of the material. A dynamic and nonlinear analysis is more fitted; if all the material characteristics are known, then we can make a good appreciation about the material and structure behavior. The analysis model by the F.E.M. can be an available model also, for the study of similar structures. Using the method described before, the problem of stress concentrators is taken into account and it is possible de determinate the maximum overload supported by a specifically structure. The experimental analysis results had fully confirmed the accuracy of the simulation.

5. References

- [1] Năstăsescu V., Pintilie Gh., Zichil V.: The study of the shear stress field in a cross section of a channeled jack by hybrid method, The 10th International Symposium on Experimental Stress Analysis and Material Testing, vol. II, Sibiu, Romania, 2004
- [2] Pintilie Gh., Zichil V., Năstăsescu V.: Analysis of the fracture conditions of an elbow under internal pressure by numerical simulation, The 10th International Conference of Fracture Mechanics, Bacau, Romania, 2004.,
- [3] Popescu D.: High pressure thick wall piping analysis (I). Fundamentals, The 10th International Conference of Fracture Mechanics, Bacau, Romania, 2004.,
- [4] Zichil, V., and co-authors: Comparison between stress and strain values obtained through analytical method and F.E.M. at bonded tubes, The 10th International Conference of Fracture Mechanics, Bacau, Romania, 2004.