Simulating of the composite cylindrical shell of the pipe of the supply pipelines based on ANSYS package

Tsyss V.G.\textsuperscript{a}, Sergaeva M.Yu.\textsuperscript{a,*}

\textsuperscript{a}Omsk State Technical University, 11, Mira Pr., Omsk 644050, Russian Federation

Abstract

The simulating of the composite cylindrical shell of the antivibration pipe based on a study of stress-strain state caused with excess pressure was carried out. Contacting interaction of the pipe shall wall in the area bordering the metal flange was conducted. The conducted finite element analysis of the stress state of the composite shell reveals the most loaded areas which represent potential danger in terms of the destruction of rubber-cord array.

© 2016 The Authors. Published by Elsevier Ltd.

Peer-review under responsibility of the Omsk State Technical University.

Keywords: composite shell, pipe, pipeline, simulating, stress-strain state, overpressure, orthotropic material, isotropic material, finite element model, the contact area, metal flange.

1. Introduction

At present antivibration pipes get wide application based on composite shells, which are designed for installation in pipelines of various systems as flexible joints in order to compensate deformations arising in the pipelines and to reduce vibrations transmitted through them. The basic requirements applied to the pipes are lack of force transfer from the fluid pressure acting on the shank bore of the pipe on the pipelines flanges or connected mechanisms.

The aim of this work is to simulate a cylindrical composite shell of the antivibration pipe based on a study of stress-strain state from the action of internal overpressure.

\textsuperscript{*}Corresponding author. Tel.: +7-381-265-2349;
E-mail address: mjsergaeva@mail.ru
2. Study subject

The basis for the creation of thrustless antivibration pipe is the theory of the equilibrium configuration of grid shells [1]. Structurally the pipe consists of two metallic flanges with which it is connected with the flange of the supply pipelines or mechanisms. At the same time a special profile of flanges allows, providing a strong fastening to pipe fittings, to achieve the desired tightness of the shank bore.

The cylindrical composite shell of the pipe has the following construction. There is a sealing rubber layer on the inner part of the shank bore which protects the bearing frame from the damage actions of the working environment and ensures tightness of the pipe wall. Next there is even number of mutually intersecting layers of the bearing frame made on the basis of synthetic or metal cords.

The bearing frame is designed to resist efforts from the pressure of working environment of the pipe. The pipe has a protective rubber coating layer outside, which makes the bearing frame safe from mechanical damages and actions of aggressive components of the environment. There are rims at the end faces of the pipe that are used for its fastening. There are metal rings of circular cross-section inside the rims and the layers of the bearing frame are wrapped around them. The rims are covered with rubber outside to provide bond strength of the covered layers of the cord and the tightness of the rims connections.

When simulating the stress state of the pipe one considers two types of the material: orthotropic - for simulating the rubber-cord (index “a”) and isotropic – for simulating rubber (index “c”). The initial data for the study are given in the table (Table 1).

Table 1. The initial data for the research.

| The task parameters          | Values |
|------------------------------|--------|
| Inner pressure, MPa         | 6.4    |
| Nominal diameter, mm        | 250    |
| The length of the examined area, mm | 1000   |
| Material of the pipe shell  | rubber cord |

The initial data for determining the specifications of the rubber cord are elastic constants of rubber and cord included in its composition. Modules of elasticity $E$ and Poisson's ratio $\nu$, are the following:

- $E_c = 7.77 \cdot 10^6$ MPa; and $\nu_c = 0.4999$ for rubber;
- $E_a = 1.1 \cdot 10^9$ MPa; and $\nu_a = 0.34$ for cord.

Coefficient of charge $m=0.65$.

We introduce the following notations:

\[
E = \frac{E_a}{E_c} = 141.57
\]

\[
G = \frac{1 + \nu_a}{1 + \nu_c} E_c^{-1} = 6.31 \cdot 10^{-3}
\]

\[
\kappa_c = 3 - 4\nu_c = 1
\]

\[
\kappa_a = 3 - 4\nu_a = 1.64
\]

\[
\chi = \kappa_c + m + (1 - m) G = 1.652
\]
\[ \zeta = 2 - m + mN_c + (1 - m)(N_a - 1)G = 2.002 \]

\[ \omega = \left(2 + (N_a - 1)G\right) \chi^{-1} - 2m(1 - G) \chi^{-1} = 0.431. \]

Modulus of elasticity in the direction of reinforcement is defined in the following way:

\[ E_i = mE_a + (1 - m)E_C = 7.17 \cdot 10^8 \text{MPa}. \]

Poisson's ratio is

\[ \nu_{12} = m\nu_a + (1 - m)\nu_c = 0.396 \]

Modulus of elasticity in isotropic plane is determined

\[ E_z = \frac{E_C}{(1 - \nu_c^2) + \nu_{12} \cdot E_C \cdot E_i} = 2.39 \cdot 10^7 \text{MPa}. \]

Shear modulus is

\[ G_{12} = \frac{1 + m + (1 - m)G}{1 - m + (1 + m)G}G_c = 1.187 \cdot 10^7 \text{MPa}; \]

\[ G_{23} = \frac{E_a}{2((1 + \nu_a)(1 + \nu_c)(1 - m))} = 7.315 \cdot 10^6 \text{MPa}. \]

Then

\[ \nu_{23} = \frac{E_a}{2G_{23}} = 0.634. \]

3. Methods

The simulating was performed with considering the pipe shell consisting of three areas (Fig. 1).

Fig. 1. Finite-element model of the cylinder shell of the pipe.

Fig. 2. The arrangement of material layers in the shell element.
Areas of section 1 have a fine grid for the simulation of the boundary disturbance. Areas of section 2 have a coarser grid, since deformation would be permanent in this zone. As the final element the membrane element has been chosen intended for simulating of thin-walled shells and the shells of the middle thickness. Moreover, this element supports the shell configuration task representing a multilayer structure. This type of construction, having a small thickness compared with its length is calculated in ANSYS software package [2].

One shows the configuration of the cylindrical composite shell of the pipe on the figure (Fig. 2). The angle of rotation of the layer relative to the shell radius is shown on the left; the material identification number is on the right (1 is rubber cord, 2 is rubber).

4. Results and discussion

The loading of a cylindrical shell of the pipe with inner pressure was performed in stages to maximum of 6.4 MPa. The next step of loading was to apply to the upper level of the shell an axial displacement of 10 mm. The final third step was to apply to the upper level the additional shear displacement of 10 mm.

The results of the first load step (with inner pressure) are shown on the figure (Fig. 3). The results of the second step of loading (with additional applied axial displacement of 10 mm) are given below (Fig. 4).

The results of the third loading step of the pipe shell (with additional applied shear to the upper level to 10 mm) are shown on the figure (Fig. 5). Distribution graphs of the main (S1, S2, S3) and the equivalent stresses according to Mises with loading of the antivibration pipe with internal pressure are shown on the figure (Fig. 6).
Of greatest interest is the study of contact interaction of the pipe shell wall in an area bordering with metal flanges (Fig. 7.). As for the physical constants for the simulated material the following values are taken:

- A1 is metal, isotropic material $E_x=2.1\times10^{11}$; $\nu=0.3$;
- A2 is rubber, hyper elastic material with the following parameters: $E_x=5\times10^6$; $\nu=0.499$; Mooney-Rivlin constants $C_1=2.93\times10^5$; $C_2=1.77\times10^5$;
- A3 is orthotropic material with the following parameters: $E_x=7.17\times10^7$; $E_y=E_z=2.29\times10^7$; $\nu_{xz}=0.396$; $\nu_{yx}=\nu_{yz}=0.634$; $G_{xy}=1.18\times10^7$; $G_{xz}=G_{yz}=7.31\times10^6$.

When partitioned into finite elements in the contact area, the following elements were used [3,4]: CONTA172 is an element for simulating two-dimensional problems with three-nodal contact elements (used to calculate the contact and sliding between the target (TARGE169) and deformable surface; TARGE169 is an element for simulating various two-dimensional surfaces connected with the surfaces presented with the element CONTA172; HYPER74 is an element for simulating solid hyper elastic structures. This element is suitable to describe virtually incompressible rubber-like materials with arbitrarily large displacements and deformations. The flat element has two degrees of freedom at each node (displacements along the axes X and Y); PLANE82 is an element providing fairly accurate results for the mixed square and triangular shapes with automatic partitioning and can satisfy the conditions of an irregular grid without significant loss in accuracy.

The contact elements are superimposed on the top of the solid-state elements and describe the boundary conditions between deformable bodies potentially in contact. A diagram of the application of the boundary conditions to the finite element model is shown on the figure (Fig. 8).

The limitation of displacement is applied along the X axis to the left side of the model as well as the limitation of displacement is applied along the Y axis on the top of the elements simulating the metal flange. The inner pressure is applied to the shank bore surface of the elements simulating the pipe. The loads $F_x=100H$, $F_y=10H$ are applied to the right of the model in the nodes.

The calculation was made considering large deformations [5]. This option is necessary to consider the incompressibility of rubber. A condition is imposed on the contact surfaces that they are in contact and the contact is not disturbed.

As a result of the applied loads and limitations, the above results are shown in the figures (Fig. 9) and (Fig. 10).
As seen from the figures, the stresses in the upper and lower layer of the shell cord of the pipe are practically identical. Figure (Fig. 9) is developed with the help of a program that creates a continuous loop by averaging the nodal stresses. The loop is determined with linear interpolation within each element using nodal values averaged at the contact nodes of the elements. The figure (Fig. 10) shows the map of the stresses developed using per element solutions with discontinuities at the boundaries of the elements. The loop is determined with linear interpolation within the element. The result averaging in the nodes is not carried out because the loop is developed independently from the adjacent elements.

The results of the contact interaction are shown in the figures (Fig. 11, 12, 13). The coefficient of friction between the outer contour of the rubber and the metal flange is assumed to be 0.5, and between the layers of the cord, rubber layers and shell cord of the pipe it is equal to 0.9. The total stress in the contact area shown on the figure (Fig. 12) differs from the contact (Fig. 11) as it considers the stresses arising in the layers slip against each other. Shearing area of the layers in the contact area of the shell and the metallic flange is shown in the figure (Fig. 13).

5. Conclusion

Thus, as a result of investigations of stress-strain state of anti-vibration pipe the following results were obtained:
1. The simulation of the composite cylindrical shell of the pipe from the internal pressure was carried out.
2. Investigations of contact interaction of the pipe shell wall in the area adjoining the metal flange were completed.
3. Finite element analysis was conducted of the stress state of the cylindrical shell of the pipe revealing the most loaded area which represents potential danger in terms of the destruction of rubber-cord array.
References

[1] V.L. Biederman, Mechanics of thin-walled structures, M., Engineering, 1977.
[2] A.S. Kravchuk, Electronic Library of mechanics and physics. Lectures on ANSYS with examples of problem solving in five parts, Minsk, 2013.
[3] J.M. Berthelot. Materiaux composites. Comportement mecanique et analyse des structures, second ed., Masson, 1996.
[4] K. Mital Subodh, L.N. Murthy Pappu, K. Goldberg Robert, Micromechanics for particulate reinforced composites, NASA TM, 107276, 1996.
[5] P. Areias, K. Matous. Finite element formulation for modeling nonlinear viscoelastic elastomers. Comput. Methods Appl. Mech. Engng, 197 (2008), pp. 4702-4717.