DOI: 10.31471/2311-1399-2019-1(11)-14-21

Investigation of strength of shaped elements of the main gas pipeline

Ya.V. Doroshenko

Ivano-Frankivsk National Technical University of Oil and Gas;
15, Karpatska Str., Ivano-Frankivsk, 76019, Ukraine

Received: 30.05.2019 Accepted: 03.06.2019

Abstract

The research has been carried out for the purpose of a complex numerical three-dimensional modeling of the stressed state of taps and tees of main gas pipelines taking into account the gas-dynamic processes occurring in these shaped elements and the temperature difference in their walls.

A 3D modeling of the elbow with a 90° angle and a reinforcing pad on the main line and the drainage of the passage line of the trunk of the main gas pipeline has been carried out. There has been studied the gas flow with 3D models of shaped elements of the main gas pipeline by means of the CFD modeling. The simulation has been performed for the equidistant tees in which the entire flow from the main stream flows into its branch. The mathematical model is based on the solution of the Navier–Stokes equation system, continuity equation, closed by a two-parametric $k-\varepsilon$ model of the Launder–Sharma turbulence with corresponding initial and boundary conditions. The simulation results are visualized in the ANSYS Fluent R18.2 Academic Postprocessor by constructing the pressure fields on the contours and in the longitudinal and transverse sections of shaped elements. The exact values of pressure at different points of the inner cavity of the shaped elements have been determined, the places of rise and fall of pressure identified. There have been performed the simulation of the temperature difference in the walls of the drainage, the trunk of the main gas pipeline in the module ANSYS Transient Thermal. The results of CFD and temperature modeling were imported into the mechanical module ANSYS Static Structural, where the finite element method was used to simulate the stressed state of the shaped elements of the main gas pipeline, taking into account the gas-dynamic processes occurring in their internal cavity and the temperature difference in the walls. The results of the simulation have been visualized by constructing a three-dimensional color fields of equivalent von Mises stresses in the tee and in the elbow. The places of the maximum equivalent stresses in the wall of the studied shaped elements have been revealed.

Keywords: *elbow, CFD modeling, Navier–Stokes equations, stress state, tee, temperature difference.*

The modern gas transmission system of Ukraine is a complex network of gas pipelines, which consists of straight sections, curves of hot (elbow) and cold bending, tees, shut-off and control valves. The greatest number of elbows, tees is contained in the piping of compressor stations, underground gas storages, gas distribution stations. The elbows contain compensators for above-ground transitions of gas pipelines, they are also in places of sharp fractures of the terrain, turns of the pipeline route. There are tees at the beginning of each branch from the main gas pipeline, in the places of jumpers between gas pipelines, at the beginning and at the end of loopings, multi-thread underwater crossings, and the like.

In the elbows, tees of the main gas pipelines the direction of movement of the product changes, which leads to a complex physical picture of the gas flow. There is a turbulent movement of the gas stream, an uneven distribution of pressure. The study of the strength of elbows, tees of main gas pipelines is complicated by the occurrence of additional stresses due to the temperature difference in their walls, uneven pressure distribution in the inner cavity of the shaped elements.

Today, such tasks can be solved by the computer simulation software package ANSYS, which provides the ability to perform multidisciplinary calculations. Using the new integrating calculation environment, ANSYS Workbench combines the strength, hydrodynamic, and temperature modules in one interface. In addition, the modern platform ANSYS Workbench allows us to simulate physical processes using 3D models built in most CAD packages.

Elbows and tees contain pipelines for various purposes (gas pipelines, oil pipelines, nitrogen pipelines, steam pipelines of nuclear and thermal power plants, pneumatic conveying pipelines, etc.). This leads the curiosity of many researchers to the study of the processes occurring in their internal cavity and the influence of these processes on the wall of the elbows.

Many computer scientists are engaged in computer modeling of hydro-gas-dynamic processes in the internal cavity of elbows, pipe tees, and the stress-strain state of their walls. Their results confirm that such software systems are an effective tool for such researches.

* Corresponding author:
ya.doroshenko@nung.eduv.ua

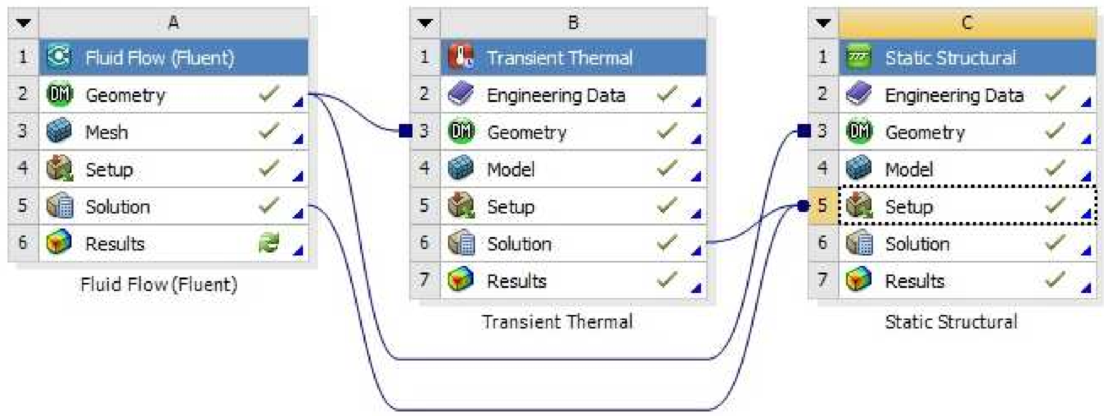


Figure 1 – The calculation scheme defined in the ANSYS Workbench calculation environment

In particular, Kumar S. and Kumar A. [1] simulated the movement of water and hydrocarbonates by elbows. The results of this simulation were imported into the ANSYS Static Structural mechanical module, where the stress-strain state of the elbows was simulated. As a result of such modeling, there was obtained pouring of pressure and flow rate in the internal cavity of the pipe elbows, diagrams of ring stresses in the wall of the elbows.

Qing-Ren W., Zhen C., Xue-Qing L., Kui W. and Lu-Yi L. [2] used numerical simulation in the Ansys software package to study the stress-strain state of a reinforced welded tee of a thermal power plant without defects and with cracks in various places of the weld and near it. Modeling was performed for different crack sizes. The calculation results were visualized by constructing three-dimensional color stress fields from which it was revealed that the greatest stresses in the tees are concentrated in the place of welding of the branch to the tee line. It was established that both the length and depth of the crack affect the value of the stress intensity factor.

Bhattacharya A. [3] calculated the stress intensity factors for reinforced tees using the finite element method. The calculation results were visualized by constructing three-dimensional color fields of the stress intensity factor from which it was found that the greatest stresses in the tees are concentrated at the point of welding of the branch to the tee line.

The existing methods for calculating the stress-strain state of shaped elements of gas mains do not take into account the uneven distribution of pressure in the internal cavity of the elbows, tees.

Therefore, the aim of the study is a comprehensive numerical three-dimensional modeling of the stress state of elbows, tees of main gas pipelines, taking into account the gas-dynamic processes that occur in these shaped elements, and the temperature difference in their walls.

Statement of the main material

The task of studying the strength of elbows, tees of gas pipelines should be solved in a three-dimensional setting. In addition, there occurs a change in the direction of movement of the product in the place of elbows and tees that leads to a complex physical picture of the gas flow. An uneven pressure distribution occurs, which affects the stress state of their wall. Also, the temperature difference affects the stress state of the wall of the elbows and tees. Therefore, it is necessary to perform a multidisciplinary calculation by combining a gas-dynamic, temperature calculation with a mechanical one.

This problem can be solved in the Finite Element Analysis ANSYS R18.2 Academic software.

The complex procedure for numerical modeling of the problem under consideration consists of six stages:

modeling of three-dimensional geometry of the walls of elbows, tees and modeling of flow geometry;

simulation of gas flow in the elbow, tee in the ANSYS Fluent module;

import of the obtained results from the ANSYS Fluent hydrodynamic module into the ANSYS Static Structural mechanical module;

simulation of the temperature difference in the walls of the elbow and tee in the Transient Thermal module for calculating thermal processes;

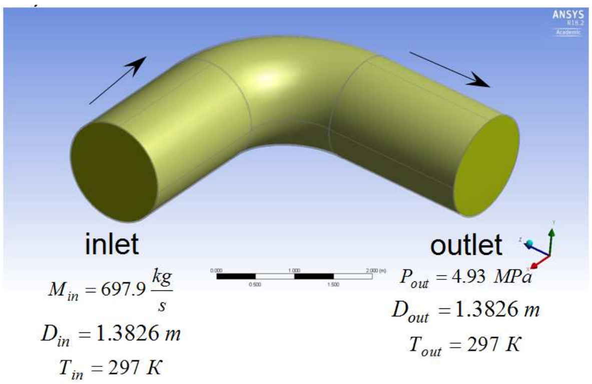
import of the obtained results from the Transient Thermal process calculation module into the ANSYS Static Structural mechanical module;

simulation of the stress state of elbows and tees in the mechanical modules of ANSYS Static Structural.

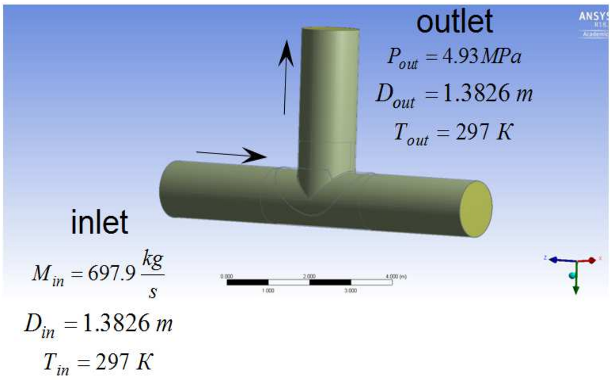
For numerical simulation of the problem under consideration, the calculation scheme shown in Figure 1 has been set in the ANSYS Workbench calculation environment.

Geometric modeling

The three-dimensional geometric model of the elbow with a rotation angle of 90° (Fig. 2 *a*) corresponds to GasTC 102-488/1 Standard [4], which are widespread in the gas industry. The outer diameter of the elbow is $1420\,mm$, a nominal wall thickness of the elbow is $24\,mm$. The outlet was modeled with adjacent pipe sections of 1 m in length, outer diameter is $1420\,mm$ and nominal wall thickness is $18.7\,mm$.



a)



b)

a) branch; b) tee

Figure 2 – Geometric models of fittings

A three-dimensional geometric model of a tee with reinforcing pads on the pipeline and the branch of the tee (Fig. 2 *b*), in which the gas moves along the line of the tee and from the line the entire flow moves into the branch of the tee, corresponds to OST 102–61 [5]. The equal tee has the outer diameter of the pipeline and branch is $1420\,mm$. The inner diameter of the pipeline and branch is $1364\,mm$. The tee was drawn with adjacent sections of the pipeline 3 m long and with an external diameter $1420\,mm$ and nominal wall thickness $18.7\,mm$. The inner diameter of the pipes is $1382.6\,mm$ and it is equal to the hydraulic diameter specified in ANSYS Fluent.

To study the strength of elbows and tees, taking into account all the loads that act on them, it is necessary to solve the connected problem of the dynamics of the gas flow movement by the branch and the stresses of the pipe walls. For this, two separate volumetric geometric models were built – for the internal cavity of the pipeline along which the gas flow moves, and for the wall of the pipeline.

Simulation of gas dynamics

The gas flow movement in ANSYS Fluent is modeled by numerically solving systems of equations describing the most general case of a moving gaseous medium. These are the Navier–Stokes equations (1), which expresses the law of conservation of momentum, (or Reynolds (2), if the flow is turbulent) and continuity (3), which expresses the law of conservation of mass

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)= =-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right)+f_i\ , \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\mathbf{u_i})+\frac{\partial}{\partial x_j}(\rho\mathbf{u_i}\times\mathbf{u_j})+\frac{\partial}{\partial x_j}(\rho\mathbf{u'_i}\times\mathbf{u'_j})= =-\frac{\partial\mathbf{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial\mathbf{u_i}}{\partial x_j}+\frac{\partial\mathbf{u_j}}{\partial x_i}\right)\right)+\mathbf{f}, \tag{2}$$

$$\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j)=0\ , \tag{3}$$

where x_i, x_j are the coordinates; t is time ; $\mathbf{u_i}$, $\mathbf{u_j}$ are speed components; ρ is the gas density; μ is molecular dynamic viscosity of the gas; $\mathbf{f}$ is a term that takes into account the action of mass forces; $\mathbf{p}$ is pressure forces; u_i are time-averaged velocity values; u'_i are the components of velocity ripple.

In ANSYS Fluent, these equations are closed by a two-parameter $k-\varepsilon$ (k is turbulent energy, ε is turbulent energy dissipation rate) turbulence model, which provides for the solution of the following equations:

turbulent energy transfer equations k

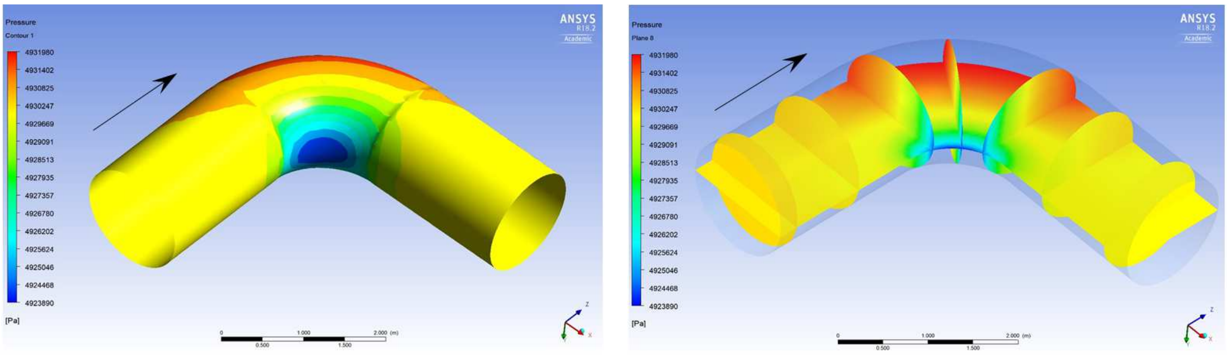
$$\frac{\partial(\rho k)}{\partial t}+\nabla(\rho uk)=\nabla\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right)+\mu_t G-\rho\varepsilon\ ; \tag{4}$$

turbulent dissipation transport equations ε

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla(\rho u\varepsilon)=\nabla\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right)+C_1\frac{\varepsilon}{k}\mu_t G-C_2\rho\frac{\varepsilon^2}{k}\ , \tag{5}$$

where u is the gas flow rate; μ_t is the turbulent dynamic viscosity of the gas; σ_k is the coefficient equal to one; G is a calculated parameter; σ_ε is a coefficient equal to $\sigma_\varepsilon=1.3$; C_1 is a coefficient equal to $C_1=1.44$; C_2 is a coefficient equal to $C_2=1.92$.

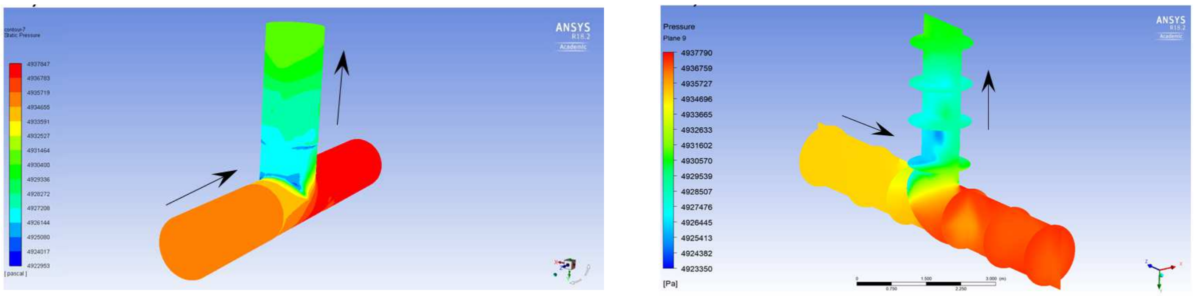
The gas flow in the elbow and tee was simulated in the ANSYS Fluent module. The Fluent–Meshing preprocessor generated a volumetric computational grid. For a better description of wall processes, a wall layer of lattices was created. There was chosen a standard two-parameter Realizable Turbulence model. For high-quality modeling of flows near the wall, the Enhanced Wall treatment function was chosen. Natural gas was



a) b)

a) on the contours; b) in the planes of horizontal longitudinal and cross sections

Figure 3 – Pressure distribution fields in the elbow



a) b)

a) on the contours; b) in the planes of horizontal longitudinal and cross sections

Figure 4 – Fields of pressure distribution in the tee in which gas moves through the tee line and the flow moves from the line to the tee branch

chosen from the ANSYS Fluent material database and assigned to the grid.

The boundary conditions, specified in the ANSYS Fluent preprocessor, are shown in Fig. 2 and Table 1.

Table 1 – Parameters of modeling gas flow in the elbow and tee of the gas pipeline

Parameter	Value
Input mass flow rate, kg/s	697.9
Temperature, K	297
Turbulence intensity, %	5
Outlet pressure, MPa	4.93
Hydraulic diameter, m	1.3826

The simulation results of the gas flow through the elbow were visualized in the ANSYS CFD postprocessor.

Pressure fields were built on the contours (Fig. 3 *a*) and in the planes of horizontal longitudinal and cross sections of the elbow (Fig. 3 *b*).

As can be seen from the pressure fields (Fig. 3, *a*, *b*), the pressure in the elbow is not evenly distributed. The flow structure in the elbow is determined by the increase in pressure in the direction from the concave to the convex side of the elbow. There is a decrease in pressure near the concave side of the elbow to 4923890 Pa and an increase in pressure near the convex side to 4931980 Pa. Such an uneven pressure distribution affects the stress state of the elbow.

As can be seen from the pressure fields (Fig. 4, *a*, *b*), the tee pressure is also not evenly distributed. In the tee line, pressure increases to 4937790 Pa at the point where the gas flows into the branch, and there is a pressure drop in the tee branch. Moreover, the maximum pressure drop is observed at the beginning of the tee branch from the opposite direction of flow of the tee line. At this point the pressure drops to 4923350 Pa. This uneven pressure distribution affects the stress state of the tee.

Modeling of the stress-strain state

Modeling of the stress-strain state of elbows and tees in the ANSYS Static Structural module was performed by the finite element method. The main ideas of the finite element method were laid down in [7] and consist in the fact that any continuous quantity, such as temperature, pressure, and displacement, can be approximated by a discrete model, which is based on many finite-continuous functions. In the general case, the continuous quantity is not known in advance, and it is necessary to determine the value of this quantity at some internal points of the section. A discrete model is very easy to built, if we first assume that the numerical values of this quantity in each inner section are known. After that, we can move on to the general case. Therefore, when constructing a discrete model of continuous magnitude, proceed as follows:

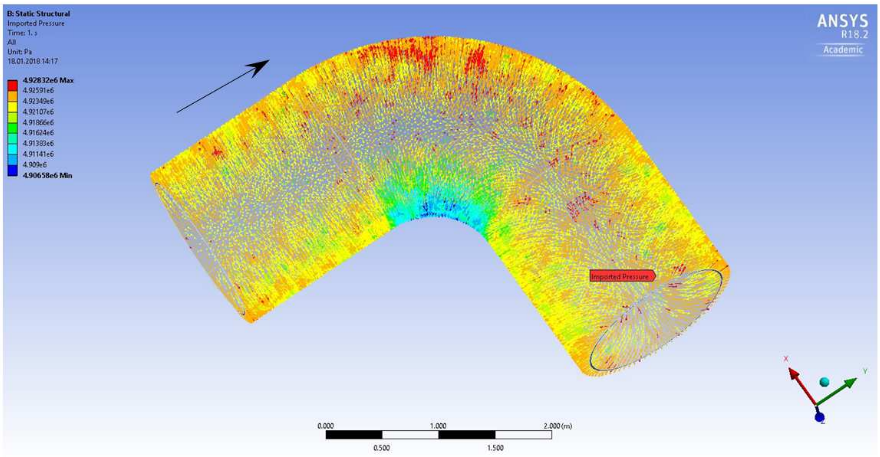


Figure 5 – Results of calculating the pressure distribution on the inner wall of the elbow imported from ANSYS Fluent into ANSYS Static Structural

in this section, a finite number of points is fixed. These points are called nodal points or nodes.

the value of a continuous quantity at each point is considered a variable, which must be determined;

the continuous domain is broken down into a finite number of sections called elements. These elements have common nodal points and collectively approximate the shape of the section;

the continuous quantity is approximated on each element by a polynomial, which is determined using the nodal values of this quantity. Each polynomial is determined for each element, but the polynomials are chosen so that the continuity of the value along the element boundaries is preserved (it is called the element function). The choice of the shape of the elements and their functions for specific tasks determines the accuracy of the approximate solution and depends on the ingenuity and skill of the engineer.

The strain function or the strain vector is expressed by the displacement function.

In case of tension, elongation of the shaft is as follows

$$\{\varepsilon\} = \left\{\frac{\partial f}{\partial z}\right\} = \frac{1}{l}\left|-1;1\right|\begin{Bmatrix}\mathbf{u_i}\\ \mathbf{u_j}\end{Bmatrix}. \tag{6}$$

We consider the expression $\frac{1}{l}\left|-1;1\right|$ as a matrix, then

$$\{\varepsilon\} = |B|\{\boldsymbol{\delta}\}, \tag{7}$$

where $\{\boldsymbol{\delta}\} = \begin{Bmatrix}\mathbf{u_i}\\ \mathbf{u_j}\end{Bmatrix}$ is the movement of the element nodes.

The stress function (stress vector) is expressed through the strain vector

$$\{\sigma\} = |D|\left(\{\varepsilon\} - \{\varepsilon_o\}\right) + \{\sigma_o\}, \tag{8}$$

where $|D|$ is the elasticity matrix (connects stresses and strains); $\{\varepsilon_o\}$ are initial strains; $\{\sigma_o\}$ are initial stresses.

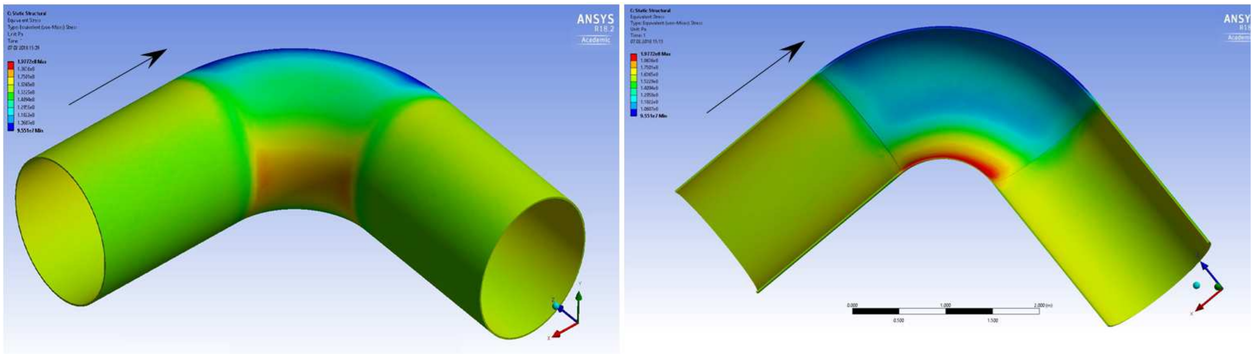
The stress state simulation of the elbow and tee was performed in the mechanical module ANSYS Static Structural. A three-dimensional geometry of their wall was imported into this module. In the database of materials of the software package, there was specified tubular steel of strength class K60 (tensile strength $\sigma_{us} = 589\ MPa$, yield strength $\sigma_{ys} = 441\ MPa$).

To take into account the influence of the uneven distribution of pressure in the internal cavity of the elbow (Fig. 3) and tee (Fig. 4) on their stress state, the results of calculating the pressure distribution on the inner wall of the elbow and tee from the ANSYS Fluent hydro-gas-dynamic module were imported into the ANSYS Static Structural mechanical module (Fig. 5). The connection of the pipeline with the soil was modeled by the Mohr–Coulomb model of elastic-plastic material integrated into ANSYS. The acceleration of gravity was applied to account for the dead weight.

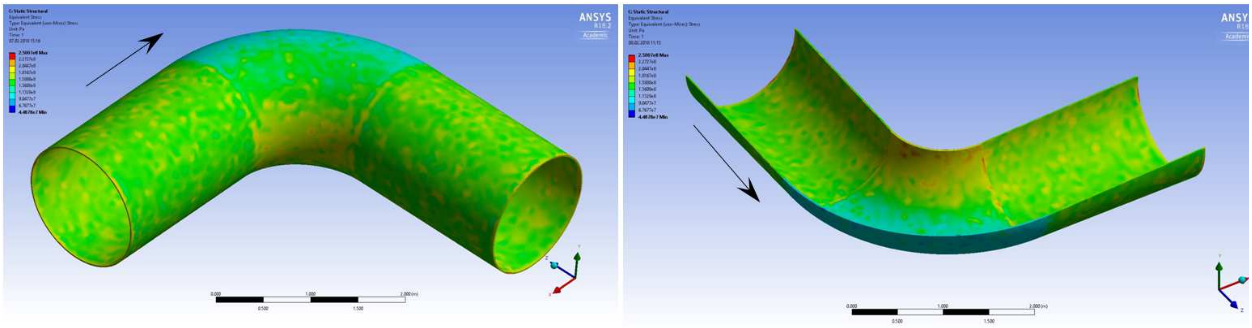
Temperature effects cause longitudinal temperature stresses at the wall of the gas pipe. The magnitude of the temperature stresses depends on the calculated temperature difference, which is taken to be the difference between the maximum or minimum possible temperature of pipeline walls during operation and the lowest or highest temperature of the pipe walls at which the calculated design of the pipeline is fixed (after laying the pipeline in a trench, fastening to the supports). For underground pipelines, the temperature difference is assumed to be ± 40 °C.

The temperature difference in the walls of the elbow and tee was simulated in the Transient Thermal module for calculating thermal processes. The characteristics of pipe steel, the temperature of the elbow wall, tee at the initial time (+20 °C) and the temperature of their wall at the final time (-20 °C) were set.

In order to take into account the influence of the temperature difference on the stress state of the elbow and the tee, results of modeling the temperature difference in the walls of the elbow were imported from the Transient Thermal module into the mechanical module ANSYS Static Structural software package.



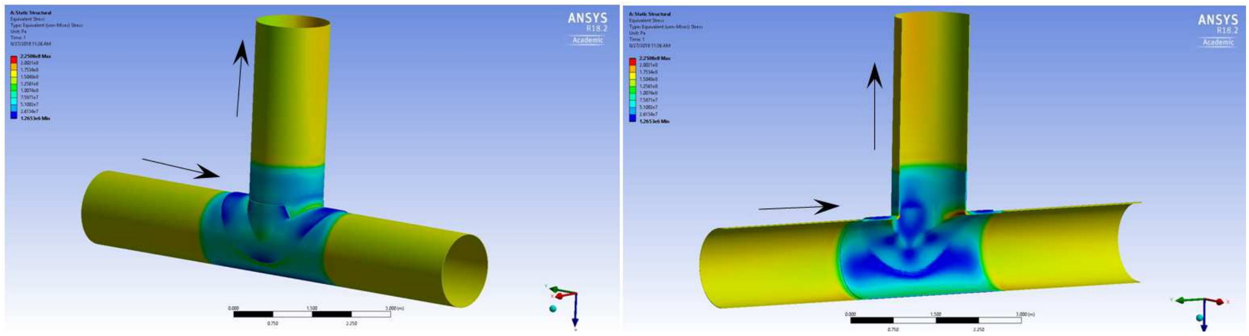
a)



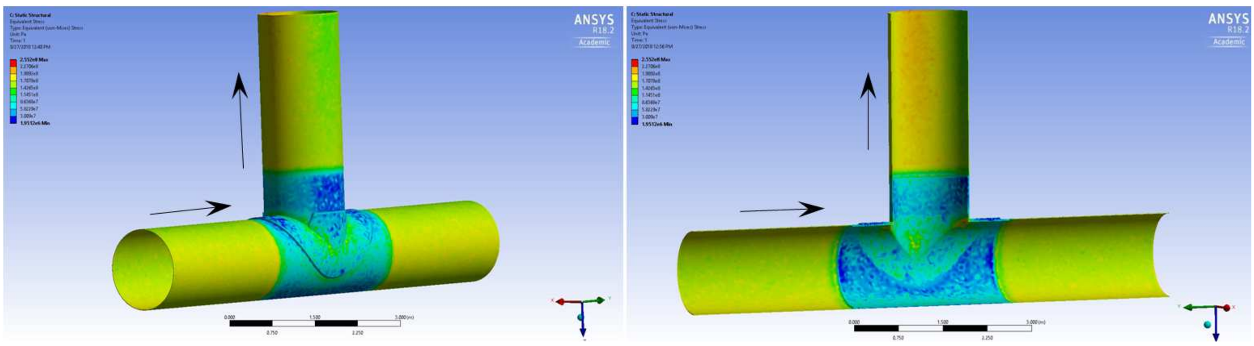
b)

a) from the effects of internal pressure; b) from the effects of internal pressure and temperature drop

Figure 6 – Distribution of the equivalent von Mises stresses in the elbow



a)



b)

a) from the action of internal pressure; b) from the action of internal pressure and temperature difference

Figure 7 – Distribution of the equivalent von Mises stresses in a tee in which gas moves through the tee line and the flow is directed from the main line to the tee branch

The results of modeling the stress state of the elbow were visualized by constructing three-dimensional colored fields of Mises equivalent stresses in the elbow from the action of only internal pressure (Fig. 6 *a*) and from the action of internal pressure and temperature difference (Fig. 6 *b*). As can be seen from the results, an uneven distribution of equivalent stresses occurs in the wall of the elbows. The maximum equivalent stresses are concentrated from the concave side of the elbow (the highest value is 197.7 MPa in the case of internal pressure only and 250 MPa in the case of internal pressure and temperature difference), the minimum with convex (the lowest value is 95.5 MPa in the case of only internal pressure and 44.9 MPa under the action of internal pressure and temperature difference). So, the minimum safety margin of the elbows is observed from the concave side. It should also be noted that the equivalent stresses on the concave side of the elbow are greater than in straight sections of the pipeline. The highest equivalent Mises stresses are 197.7 MPa on the concave side of the elbow in the case of internal pressure only and 250 MPa under internal pressure and temperature difference.

The results of modeling the stress state of a tee with reinforcing pads in the pipeline and branch were visualized by constructing three-dimensional colored fields of equivalent von Mises stresses in the tee from the action of only internal pressure (Fig. 7 *a*) and from the action of internal pressure and temperature difference (Fig. 7 *b*). As can be seen from the results, an uneven distribution of equivalent stresses occurs in the wall of the tees. The maximum equivalent stresses in the tee are concentrated at the point of connection of the branch to the main of the tee, where there is no reinforcing lining on the branch of the tee (the highest value is 225.1 MPa in the case of only internal pressure and 255.2 MPa in the case of internal pressure and temperature difference). So, the minimum safety margin of the tees is observed at the point of connection of the branch to the tee line, where there is no reinforcing lining at the branch of the tee.

Conclusions

A step-by-step algorithm is proposed for three-dimensional modeling of the stress state of elbows, tees of main gas pipelines, taking into account the gas-dynamic processes that occur in their internal cavity and the temperature difference in the walls.

Having performed CFD modeling of gas-dynamic processes, it was found that there is an increase in pressure in the elbows of the main gas pipelines in the direction from the concave to the convex side of the elbow. Pressure increases in the line of the equal tee at the place of overflow of the entire gas stream in the branch, and the pressure drops in the branch of the tee.

Simulation of the stress state showed that the minimum safety margin of the elbows of the main gas pipelines is observed from the concave side of them, and the minimum safety margin of the equal tee of the reinforced tees is at the point of connection of the branch to the main of the tee, where there is no reinforcing lining on the branch of the tee. The equivalent von Mises stresses of the considered shaped elements do not exceed the permissible ones at operating parameters.

Such results provide with opportunities for a full and comprehensive study of the strength of branches and tees of gas pipelines, taking into account their actual technical condition and operational parameters.

References

[1] Kumar, S & Kumar, A 2017, 'Computational fluid dynamics simulations of pipe elbow flow', *International journal of professional engineering studies*, vol. 9, iss. 2, pp. 32–42.

[2] Qing-Ren, W, Zhen, C, Xue-Qing, L, Kui, W & Lu-Yi, L 2016, 'Cracking failure analysis of T-type welded tee in waste heat boiler', *Advances in Engineering Research*, vol. 103, pp. 427–432.

[3] Bhattacharya, A 2011, *A finite element based study on stress intensification factor (SIF) for reinforced fabricated tees*, *NAFEMS Word Congress*, 23–26 may 2011, Boston, NAFEMS, p. 66.

[4] Specifications 2005, *Connecting parts for gas mains up to 9.8 MPa (100 kgf/cm^2)*, the State Standard GazTU 102-488/1–05 2005, OJSC Trubodetal.

[5] Industry standard 1981, *Details of steel pipelines welded up to 10.0 MPa (100 kgf/cm^2)*, OST 102-61–81, Ministry of Construction of Oil and Gas Enterprises.

[6] Squires, K, Eaton, J 1990, 'Particle response and turbulence modification in isotropic turbulence', *Phys. Fluid*, vol. 2, no. 7, pp. 1191–1203.

[7] Gallagher, RH 1975, *Finite element analysis: fundamentals*, Prentice-Hall, New York, 416 p.

УДК 622.692.4

Дослідження міцності фасонних елементів магістрального газопроводу

Я.В. Дорошенко

Івано-Франківський національний технічний університет нафти і газу;
вул. Карпатська, 15, Івано-Франківськ, 76019, Україна

Дослідження виконано з метою комплексного числового тривимірного моделювання напруженого стану відводів, трійників магістральних газопроводів з урахуванням газодинамічних процесів, які відбуваються в цих фасонних елементах, температурного перепаду в їх стінках.

Здійснено 3D моделювання відводу з поворотом на кут 90º і підсиленого накладкою на магістралі і відводі рівнопрохідного трійника магістрального газопроводу. CFD моделюванням досліджено рух газового потоку 3D моделями фасонних елементів магістрального газопроводу. Моделювання виконано для рівнопрохідного трійника в якому увесь потік з магістралі перетікає у його відвід.

Математична модель базується на розв’язанні системи рівнянь Нав’є–Стокса, нерозривності, замкнених двопараметричною $k-\varepsilon$ моделлю турбулентності Лаундера–Шарма з відповідними початковими і граничними умовами. Результати моделювання візуалізовано в постпроцесорі ANSYS Fluent R18.2 Academic побудовою полів тиску на контурах та у повздовжньому і поперечному перерізах фасонних елементів. Визначено значення тиску в різних точках внутрішньої порожнини фасонних елементів, виявлено місця підвищення та падіння тиску.

В модулі ANSYS Transient Thermal виконано моделювання температурного перепаду в стінках відводу, трійника магістрального газопроводу. Результати CFD та температурного моделювання імпортовано у механічний модуль ANSYS Static Structural, де методом скінченних елементів виконано моделювання напруженого стану фасонних елементів магістрального газопроводу з урахуванням газодинамічних процесів, які відбуваються у їхній внутрішній порожнині, та температурного перепаду в стінках.

Результати моделювання візуалізовано побудовою тривимірних кольорових полів еквівалентних напружень за Мізесом у трійнику та відводі. Виявлено місця максимальних еквівалентних напружень в стінці досліджуваних фасонних елементів.

Ключові слова: *CFD моделювання, відвід, напружений стан, рівняння Нав’є–Стокса, температурний перепад, трійник.*