International Journal of Research Studies in Computer Science and Engineering (IJRSCSE)
Volume 2, Issue 8, August 2015, PP 16-24
ISSN 2349-4840 (Print) & ISSN 2349-4859 (Online)
www.arcjournals.org

Application of FEM in the Fluid Flow in the Pipeline

Helena Kravarikova

Faculty of materials science and technology in Trnava,
Slovak university of technology in Bratislava,
Paulínska 16, 917 24 Trnava, Slovakia.
helena.kravarikova@stuba.sk

Abstract: *Fluid flow conduit is a very complex one-dimensional motion of a viscous liquid. This movement cannot be generally analytically solved on the basis of currently available physical evidence. Some mathematical calculations are too complex, sometimes intractable. Hence the need for the development of numerical methods that are able to analytically solve intractable problems. Currently, the most commonly employed method is the finite element method (FEM). An integral part in various areas of research and development is the use of numerical simulations using FEM. Troubleshooting can be further examined from different perspectives using numerical simulations. With the rapid development of computer technology, numerical analyzes using FEM can process much faster, more precise monitoring of the examined parameters with the possibility of alternative solutions. Using mathematical models can describe the behavior of liquids in the process flow or in relative peace. The result of FEM application on fluid flow in a piping system analysis is obtained from the solution of the problems of numerical simulation using ANSYS. The paper is focused on the implementation of numerical fluid flow process in the application of technical devices to solve real problems.*

Keywords: *FEM, numerical simulation, fluid task, velocity fields, pressure fields.*

1. Introduction

As the liquid flow pipe has considerable experience in technical applications. Therefore, the three-dimensional fluid flow examines the introduction of certain simplifying assumptions. Simplifying assumptions will allow to the address the role of accuracy, satisfactory in technical applications.

Simplistic assumptions are:
• pipe length is many times greater than the transverse dimensions of ducts,
• liquid flows in a pipe axis w at a rate which is the ratio of flow rate Q and the pipe cross-section and perpendicular to the pipe axis, $w = Q / A$,
• speed w is constant throughout the cross-section A, the vector is parallel to the axis of the pipes in the appropriate place.
• A cross-sectional changes hydrostatic pressure, as a basis for considering the pressure in the pipeline axis.

When solving problems, it is necessary to respect the two laws of nature, the conservation of mass law and energy conservation law. The introduction of fluid flow in the pipe axis receive one-dimensional flow model. While respecting the established assumptions and conservation laws of energy and mass, then the interpretation of the one-dimensional and three-dimensional flow is the same. We can be applied to the fluid flow in various technical installations.

2. Theoretical Conditions of the Phenomenon

The technical equipment at fluid flow it must be assumed the validity of conservation of mass and law of energy conservation. Navier-Stokes equation expresses the balance of power in the real fluid when disturbed the balance of forces acting on the fluid element. The consequences of breaching the balance of forces on the element fluid flow passage of liquid under the force of Newton's second law. Therefore, the inertia force is equal to the sum of the mass, pressure and friction forces. Then the balance of forces acting on the element of fluid in the fluid flow can be written in vector form as follows:

$\mathbf{F}_m + \mathbf{F}_p + \mathbf{F}_t = \mathbf{F}_z = m.\mathbf{a}$ [N] (1)

Wherein:

$\mathbf{F}_z$ [N] – vector Inertia force
$\mathbf{F}_m$[N] – vector Mass forces
$\mathbf{F}_p$ [N] – vector Pressure forces
$\mathbf{F}_t$ [N] - vector Friction forces
$\mathbf{a}$ [m.s^{-2}] – vector acceleration
m [kg] - Weight

The flow of real fluids causes the acceleration vector. The normal and the tangential component the acceleration caused by the formation of normal and shear stress in the flow of real fluids.

The resulting acceleration of the fluid element can be represented by the sum of the local and convective acceleration in the form [1, 2, 3]:

$$\mathbf{a} = \frac{\partial \mathbf{w}}{\partial t} + \mathbf{w}\ grad\ \ w. \qquad [\mathrm{m.s^{-2}}] \qquad (2)$$

Mass force is defined:

$$\mathrm{d}F_m = R.\mathrm{d}m = \rho.R.\mathrm{d}V = \rho.R.\mathrm{d}x.\mathrm{d}y.\mathrm{d}z \qquad [\mathrm{N}] \qquad (3)$$

The pressure force F_p applied to the surface of a selected rectangular elementary volume in the three perpendicular directions of the coordinate system. The pressure exerted on the surface can be considered constant, because the surface of the elemental volume is infinitely small. The resulting pressure force acting on the selected element of the fluid in the *x*-direction is defined by:

$$\mathrm{d}F_{px} = -\frac{\partial p}{\partial x}\mathrm{d}x\,\mathrm{d}y\,\mathrm{d}z. \qquad [\mathrm{N}] \qquad (4)$$

By analogy we get pressure forces acting in the direction of the coordinate axes y and z. Frictional force can be expressed tangential stress applied on an area:

$$F_\tau = \tau.A = \eta\frac{\mathrm{d}w}{\mathrm{d}y}A, \qquad [\mathrm{N}] \qquad (5)$$

Wherein:

η	[N.s.m^{-2}]	- dynamic viscosity
A	[m^2]	- area
$\frac{\mathrm{d}w}{\mathrm{d}y}$	[-]	- velocity gradient

Of the balance of forces acting on the elementary volume of liquid is Navier - Stokes equations, we can write it in vector form [1, 2, 3]:

$$\mathbf{R} - \frac{1}{\rho}grad\ \ p + \nu\,\Delta\,\mathbf{w} = \frac{\partial \mathbf{w}}{\partial \mathrm{t}} + \mathbf{w}\ grad\ w. \qquad [\mathrm{m.s^{-2}}] \qquad (6)$$

If we neglect shear stress, then we talk about so called model with zero viscosity flow without friction. Then equation (1) in vector can be written in the form:

$$\mathbf{F}_m + \mathbf{F}_p = \mathbf{F}_z. \qquad [\mathrm{N}] \qquad (7)$$

Effect of the unit forces the selected element flowing fluid can be written in vector form: $\mathbf{R} - \frac{1}{\rho}grad\ \ p = \frac{\partial \mathbf{w}}{\partial t} + \mathbf{w}\ grad\ w$ [m.s^{-2}] (8)

Equation (8) represents the Euler equations of hydrodynamics in vector form. The Euler equations of hydrodynamics have represents the relationship between the unit mass, pressure and inertial forces acting on the elementary particles of liquid [1, 2, 3].

From Euler equations of hydrodynamics, assuming steady flow of fluid in the gravity field, then applied per mass force has the effect of gravitational force. Based on these preconditions we can express the law of conservation of energy Bernoulli equation. Means a relation between kinetic, pressure and potential energy on the streamlines of an ideal fluid, equation (9):

$$e = gh + \frac{p}{\rho g} + \frac{w^2}{2g} = \text{constant} \qquad [\text{J.kg}^{-1}] \qquad (9)$$

But flow of fluid friction cannot be neglected in those places where there is a large velocity gradient. In that event formed resistances force in the flow of the real fluids. To overcome the resistance forces flowing fluid runs out of the energy flow. This energy loss is the work of friction per unit mass of flowing liquids. The loss energy is expressed by Weisbach relations for local and length loss in the form:

$$e_{z\,m} = \zeta \frac{w^2}{2} \qquad [\text{J.kg}^{-1}] \qquad (10)$$

$$e_{z\,l} = \lambda \frac{L}{d}\frac{w^2}{2} \qquad [\text{J.kg}^{-1}] \qquad (11)$$

Bernoulli's equation for the flow of real fluid jet fiber has the form:

$$e_1 = e_2 + e_z = g\,h_1 + \frac{p_1}{\rho} + \frac{w_1^2}{2} = g\,h_2 + \frac{p_2}{\rho} + \frac{w_2^2}{2} + e_{z\,m} + e_{z\,l} \qquad [\text{J.kg}^{-1}] \qquad (12)$$

Conservation of mass defined by the continuity equation in the form of volumetric flow:

$$Q = w\,A \qquad [\text{m}^3.\text{s}^{-1}] \qquad (13)$$

3. Defining the Phenomenon and Conditions for the Solution

Y-shaped piping is shown in Fig. 1. The branch lines are symmetrical about the x-axis angle α = 60 °. The pipe diameter is D = 200 mm, d = 150 mm. Pipe diameter D steady fluid flows through the volume flow Q = 0.18 m3.s-1. Consideration must be given to the division of the flow Q to the flow rate in the two branches of the same. Energy dissipation can be neglected. Our task is to gain speed and pressure field of fluid flowing in the pipeline.

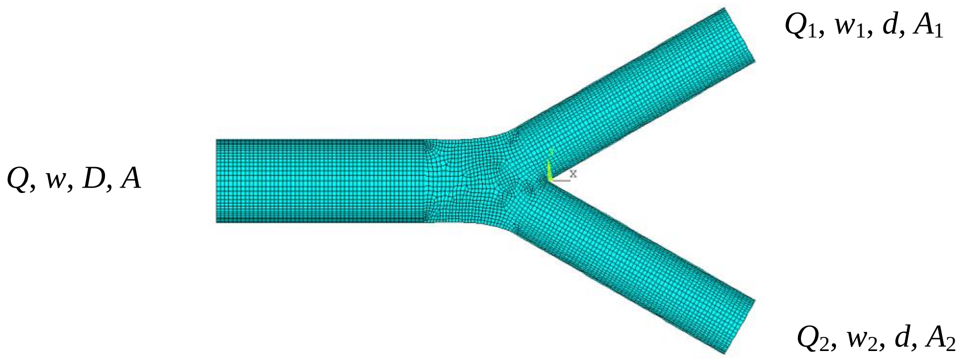

Fig1.*Scheme Y-pipe with generating network*

3.1 Conditions for the Solution Tasks Numerical Simulation

Solving tasks will be carried out by means of computer equipment by FEM using ANSYS. In our case we will deal with the fluidic tasks using numerical simulations. For a fluidic role of analytical solutions, the conservation of mass and using mathematical relation of theoretical introduction, the calculated fluid flow rates in pipes. The results are written in Table 1.

Table1.*The calculated value of the fluid flow in the pipe*

D 0.2 [m]	***w*** [m.s^{-1}]	***d*** 0.15 [m]	***w*** [m.s^{-1}]	***d*** 0.15 [m]	***w*** [m.s^{-1}]
section ***A***	5.73	section A_1	5.09	section A_2	5.09

To calculate the speed and pressure fields, numerical simulation of fluid flow it was a Y-shaped pipe created a geometric model, exactly as entered geometric parameters. The geometric model is been divided into a finite number of elements, Fig. 1. For solving numerical simulation by FEM, in addition to the geometric model is required mathematical model for the type characteristic of the phenomenon. The mathematical model for solving tasks of a fluid flowing fluid specified in a theoretical introduction to the phenomenon. A very important step is to properly define initial and boundary conditions. When we are considering the tasks a fluidic flow rate of fluid on the walls of the pipe equal to zero, as the particles flowing fluid on the walls of the pipe is of braked. Not we are considering the energy dissipation. The type of numerical experiment was solved with turbulent flow incompressible fluid as unsteady adiabatic task. Conditions solutions speed and pressure field numerical simulation was solved for different alternatives direction of flow of the flowing fluid and a pressure of 200 kPa at the beginning of the entrance to the pipe. For the numerical experiment, we used most liquid, an water density 998 kg.m-3 and the dynamic viscosity 10.20.10-4 Pa.s at temperature 20 ° C and atmospheric pressure of 101.3 kPa.

4. Results of the Numerical Experiments

Steady flow in this pipe, the solution speed and pressure fields using numerical simulation, achieve more steps. Reported results for different alternative solutions they are obtained at steady flow process.

Fig. 2 graphically shows the velocity fields with the direction of the jet schematically indicated on the image details. With a pressure of 200 kPa is contemplated the entrance to the pipe diameter D. For the following conditions fluid flow in the Y-pipe we reached evenly distributed velocity field with an entry speed of 5.73 ms-1 along the entire length of the horizontal pipe diameter D. The maximum calculated speed of 8.91242 ms-1 is obtained in the pipe of diameter D, near the wall of the distribution pipe at the flow of liquid into the two branches of the cross sections A1 and A2. The fluid input to the pipelines, the quasi-same speed and the speed fields are symmetrical about the x axis.

The velocity Field of the jet pipe in the shape of Y is represented by vectors, together with details of the flow distribution of the fluid, is in Fig. 3. The graphic representations observed steady flow direction of the liquid to be divided into two symmetrical branches of the pipeline.

Fig. 4 graphically shows an even distribution of pressure across the field of horizontal pipes with a diameter D and two-pipe cross-section A1 and A2. Distribution is uneven stress field is in the area of the flow distribution into the various branches of pipeline. Pressure field corresponds to the velocity distribution in a given section of pipe. Maximum pressure rating at is the point of a sharp to a branching manifold and the Y-nearby. In this part of the fluid jet impinges on the pipe wall.

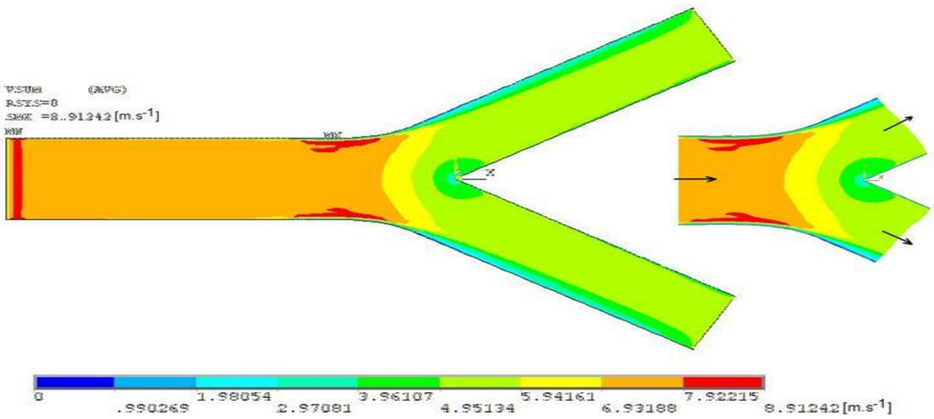

Fig2.*Velocity fields in the steady process fluid flow*

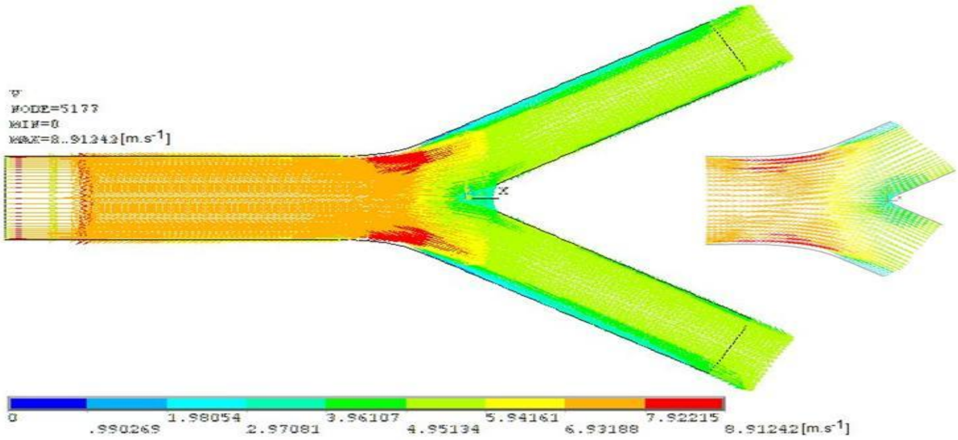

Fig3.*Velocity fields in the steady process in the form of vectors*

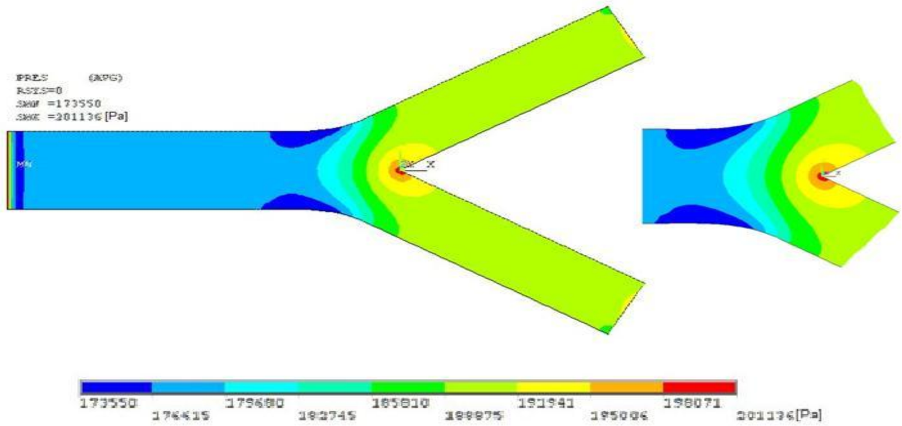

Fig4.*Pressure field at steady process fluid flow*

Fig. 5 graphically illustrates the velocity field of the opposite direction of the jet as in the previous section. The direction of fluid flow is indicated on the details. Pressure of 200 kPa considered to enter line with the cross section A2, the bottom of pipeline. The pressure is expressed fluid into the pipe. In the second branch of the Y-pipe cross section A1 considered only to atmospheric pressure. In this part of pipeline fluid flow occur under the action of gravity. For the conditions of the fluid flow in the Y-pipe we obtained velocity Field spread evenly distributed over the length of a horizontal pipe and the section height A. The maximum calculated speed of 18.7888 ms-1, is achieved in the influent liquid in horizontal pipes with a diameter D, in the vicinity of the pipe wall. The fluid input to the pipelines, the quasi-same speed 5.09 ms-1. Velocity fields in the pipe with a cross section of A1 and A2 are stabilized with different speed values corresponding to the given terms of the settlement.

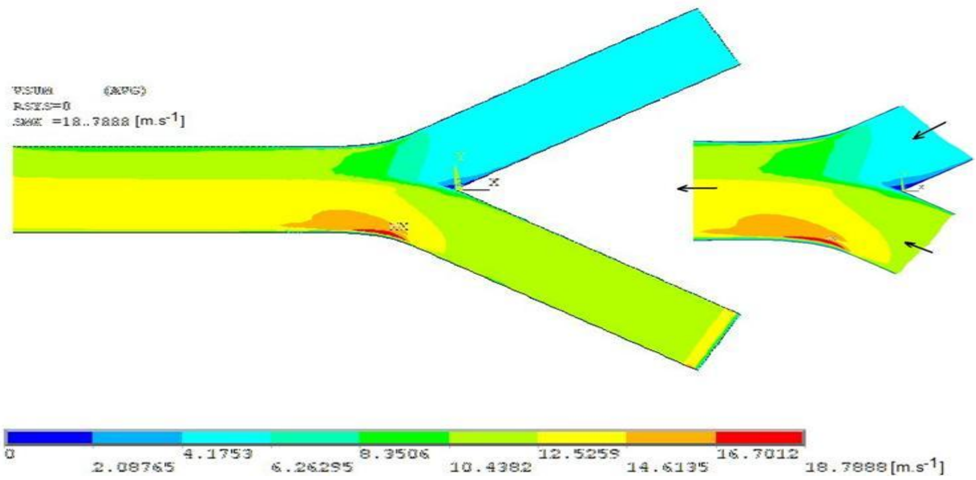

Fig5.*Velocity fields in the steady process fluid flow*

Field of the jet velocity in the pipe in the Y vector is displayed together with details joining the liquid flow shown in Fig. 6. The graphic illustration can be observed a steady direction of the fluid flow in each branch of the pipeline, which is connected to a single stream in pipe diameter *D*.

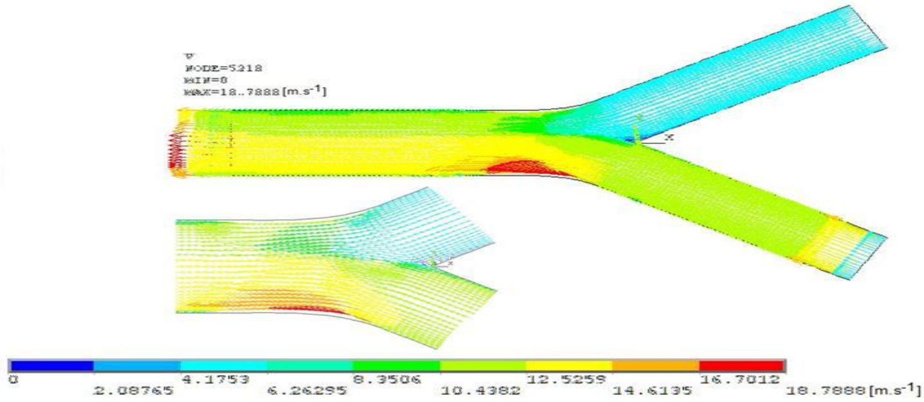

Fig6.*Velocity fields in the steady process in the form of vectors*

Pressure field is shown in Fig. 7. The graphic illustration of the flow conditions of the fluid can be observed that in the two the cross section of the branch pipes A1 and A2 is a spread evenly pressure field with a maximum pressure of 111 kPa. In areas linked to pipe diameter D is the pressure field distribution is uneven, and there is a drop in pressure.

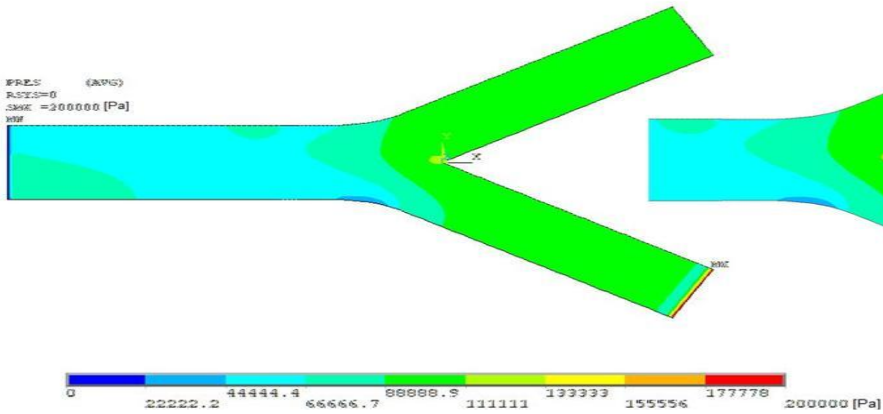

Fig7.*Pressure field at steady process fluid flow*

Fig. 8 is shown velocity fields with the direction of the jet stated on details. The fluid enters line in the upper part the strand cross section A1 and the pressure at the pipeline of 200 kPa. With a pressure of 200 kPa contemplated the output from the horizontal pipe diameter D. The second branch pipeline Y-shaped cross-section A2 considered in the outflow liquid only with atmospheric pressure and gravity. For the conditions we obtained steady speed field along the length of the horizontal pipe diameter D and top of the line a cross-section A1. The maximum rate of 22.5916 ms-1, was reached in the pipe cross-section A2, right after the injection of fluid into the lower branch of the pipeline. Fluid enters the pipeline cross-section A1 with uniformly distributed speed of 5.09 ms-1 and the output from the horizontal pipe diameter D uniformly distributed speed.

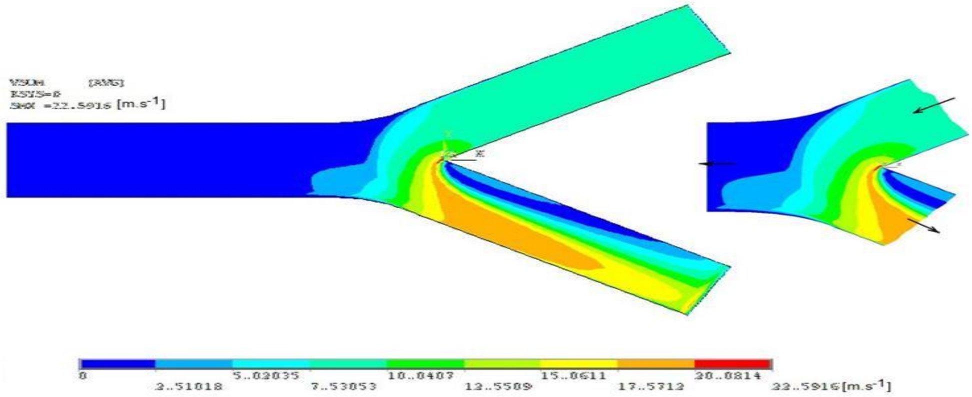

Fig8.*Velocity fields in the steady process fluid flow*

Creating turbulence occurs at on the coupling point pipelines, and the direction of the inlet fluid in the pipe cross-section A_2, Fig. 9. The detail of Fig. 9 shows the flow direction of the fluid flow and a turbulent flow.

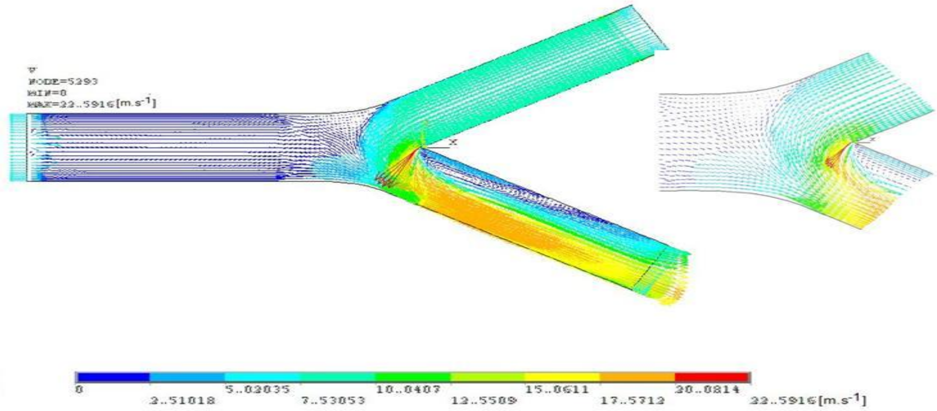

Fig9.*Velocity fields in the steady process in the form of vectors*

Pressure field in Fig. 10 is spread evenly in the horizontal section of the pipe diameter D and a branch with section A1. In this case, the pressure field is quasi-constant value of 200 kPa. In the area of distribution pipeline to individual branches across the underside of the branch pipeline cross-section A2 there is an uneven stress field and pressure drop. Illustrated pressure field is adequate velocity field being formed for the conditions fluid flow in the pipeline.

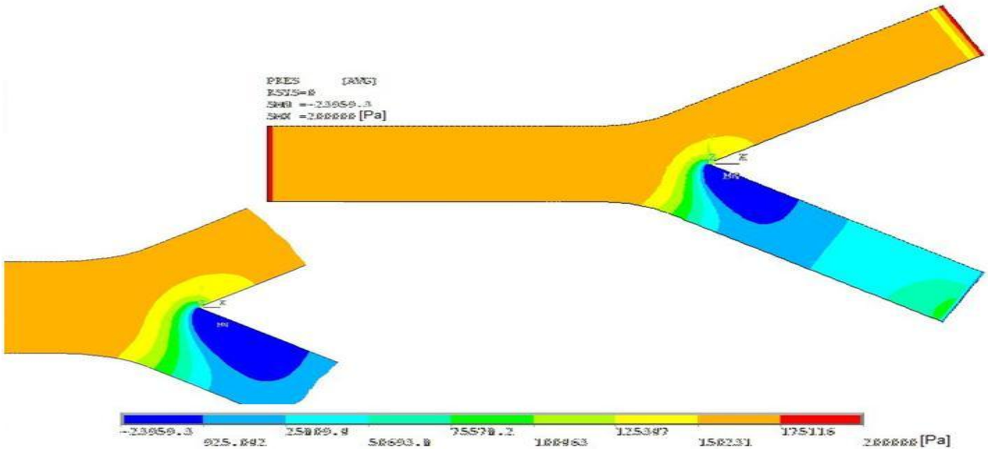

Fig10.*Pressure field at steady process fluid flow*

Fig. 11 is shown velocity fields with the direction of the jet stated on details. The Y-pipe, we obtain even the forming velocity field over the length of a horizontal pipe, the diameter D of an entry speed 5.73 ms-1 and a pressure of 200 kPa and also the upper-pipe cross section of A1 of the input speed of 5.09 ms-1 a pressure of 200 kPa. At the outlet in the bottom of pipeline cross-section A2 in the fluid outlet operates atmospheric pressure and the earth's gravity. In this part of the pipeline velocity field it is uneven and maximum speed has a value of 17.7151 ms-1.

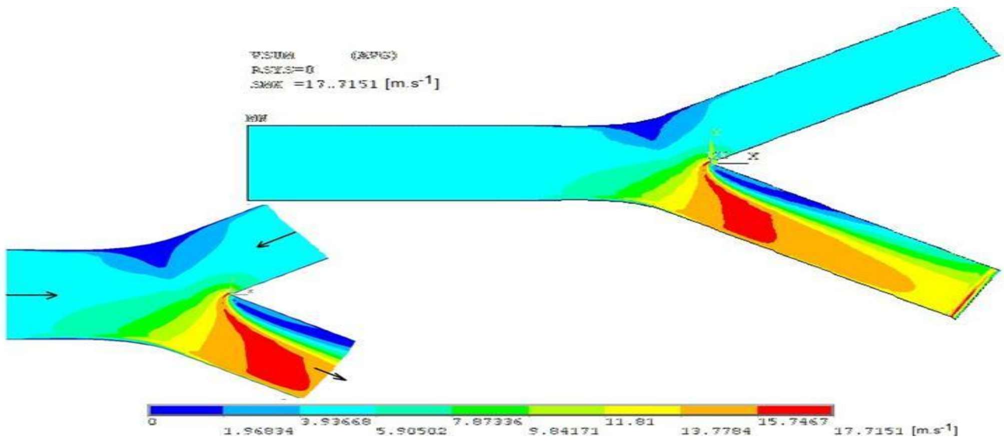

Fig11.*Velocity fields in the steady process fluid flow*

Velocity fields represented by the vectors of the liquid flow is steady in Fig. 12.Detail of the figure shows the formation of turbulent flow and the formation of the vacuum in the areas of local subdivision pipeline.

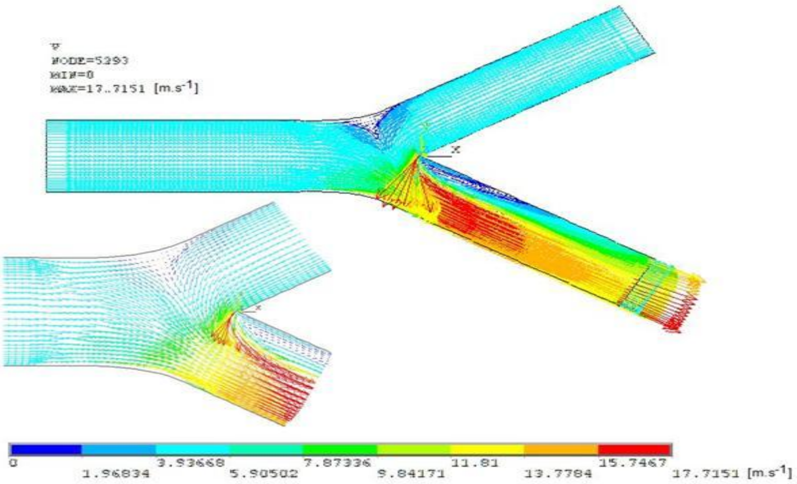

Fig12.*Velocity fields in the steady process in the form of vectors*

For this alternative fluid flow is even distribution of pressure field with a maximum pressure of 216 912 Pa, throughout the part of the horizontal pipe diameter D and also in the upper-pipe cross-section A1, Fig. 13. In the lower-pipe cross-section A2 is unevenly distributed pressure field, which corresponds to an uneven velocity distribution in a given section of pipe.

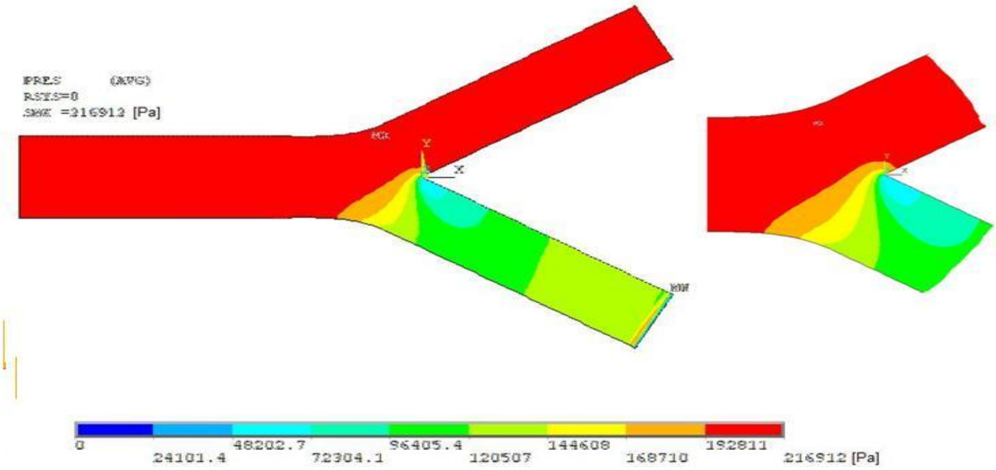

Fig13.*Pressure field at steady process fluid flow*

5. Evaluate the Results

Using numerical experiments was applied solution for the selected alternative definition of the tasks for the flow of fluid in the pipe Y. In solving tasks creating a geometric and mathematical model the numerical simulation of using FEM depends on various factors. It is important to apply the correct initial and boundary conditions of the phenomenon ensure the attainment the desired results of numerical simulations. In our case, it was using numerical simulations to get the results of the pressure distribution and velocity field of alternatives solved tasks and evaluate the nature of flow with maximum and minimum speed and the most stressed areas of high pressure. For solving other alternatives were given boundary conditions, so the speed and pressure fields have different layout maximum and minimum values.

6. Conclusion

So we can extend the life of technical equipment and prevent the financial and material losses, it is necessary that we know all the best load of the in operation on the device. Therefore, we must know how to behave during fluid flow in technical equipment. This knowledge can affect the durability, cost and efficiency of the device. Therefore need for the development of numerical methods by which the analytically intractable problems can be solved. Problem solving can be more examined from different perspectives using numerical simulations. With the rapid development of computer technology, the numerical analysis of FEM can process more quickly, more accurately and with the possibility to change the endpoints. The object of our interest can be more examined from different perspectives using numerical simulations. Numerical simulation used as an aid to monitor the real system. We can be considered as a specific tool which can model the real system in a virtual environment. Using simulations, we can test any variants flexibly respond to the identified tasks. Therefore, it is an integral part of research and development in various areas of numerical simulations using FEM.

REFERENCES

[1] Taraba B., Behulova M., Kravarikova H.: Mechanika tekutín Termomechanika. Bratislava : STU Bratisva, 2004. s. 10. ISBN 80-227-2041-0.

[2] Drabkova, S.a kolektív: Mechanika tekutín. Ostrava: VŠB - TUO, 2007. ISBN 978-80-248-150-4.

[3] Janalik, J.: Hydrodynamika a hydrodynamické stroje. Ostrava: VŠB - TU, 2008.