NUMERICAL ANALYSIS OF FLUID FLOW IN A SYRINGE – TEST TUBE

Wiesław Krasoń, Jacek Łazowski, Jerzy Małachowski

Military University of Technology
Department for Mechanics and Applied Computer Science
Gen. Sylwestra Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 22 6839683, fax: +48 22 6839355
e-mail:wkrason@wat.edu.pl, jlazowski@wat.edu.pl, jerzy.malachowski@wat.edu.pl

Waldemar Pakszys

Military Medicine Institute
Szaserow Street 128, 04-141 Warsaw 44, Poland
tel.: +48 22 681 76 66, fax +48 22 681 66 94

Abstract

Numerical research in the field of collaboration of basic components and entire constructional solutions is much cheaper than managing a goods production and testing them in actual conditions. Numerical research allows determining essential modifications of a product being prepared from the functional, utilitarian and technological point of view. Considering the problem to be solved, the blood flows through the area limited by the sides of a piston, syringe and needle were analyzed. Pressure arises after the initiation of the piston's move and the blood flows as a result of the difference in pressures. The blood flows from th e areas with higher pressure to the ones with lower pressure. The numerical research focused on analysis of the boundary conditions influence on the arrangement of the velocity and blood pressure in a syringe. In order to calculate the blood flow through a syringe the researchers used a model that allows the flow of adhesive fluids - subst ances with internal friction. Navier-Stokes' equations were used to describe adhesive fluids move mathematically. These equations allowed describing non-stationary flow of incompressible fluids, i.e. including no volumetric changes, however, including figural changes. In last analyzed case, i.e. with flow orifice, fluids dam up in front of the orifice and then flow into the next chamber just after the orifice. The fluids flow to the chamber as a piled up stream. Then, the fluids hit internal bend sides and rotate again in front of the needle orifice en try. Considering fluids parameters, vorticity process improves fluid mixing process.

Keywords: *syringe, blood, flow, simulation, vorticity*

1. Introduction

In the system being under consideration, the blood was flowing through the area between the sides of the piston and syringe. Pressure arises after the initiation of the piston's move and the blood flows as a result of the difference in pressures. Blood flows from the areas with higher pressure to the ones with lower pressure. Stream lines appear in the area of blood flow. This characterizes places with the same velocity. The smallest velocity has the liquid area that is directly edged with a partition in the blood flow process. If a cross-section is changed, we experience the change of velocity gradient between particular areas of the adhesive fluid. In this case, i.e. if a cross-section is changed and there is a laminar flow, researchers assume that the energy is constant. With turbulent flow a portion of energy might be lost, which must be included in the energy equilibrium. In order to compute the blood flow through a syringe the researchers used a model that allows the flow of adhesive fluids. Adhesive fluids are substances with internal friction, which, if motionless, cannot balance shear stress and at the same time take approximately constant volume and form the area [1-4]. It is possible to use Navier- Stokes' equations to describe adhesive fluids move mathematically.

The mentioned equations used in FE approach were applied in studied cases to show the influence of the orifice introduction into the syringe on the flow parameters. The authors based on performed computations analyzed such parameters like a pressure change and a fluid velocity.

2. Basic equations describing incompressible fluids flow

The basic equation for incompressible fluid flow in symbolic form can be presented as follows [1-4]:

$$\rho\frac{\partial\vec{u}}{\partial t}+\rho\left(\vec{u}\cdot\nabla\right)\vec{u}=\rho\vec{f}-\operatorname{grad}p+\mu\Delta\vec{u}, \tag{1}$$

where:
ρ - fluid density,
μ - viscosity,
t - time,
p - pressure,
u - velocity vector,
f - intensity field of mass force (e.g. gravity forces).

It can be proved that the stress in each point of the space filled with adhesive fluids is determined by numerical value of six, not nine stress components. Thus, shear convergent stresses on the same edge are equal (there is a symmetry of shear stresses). The basic assumption which allows quantity connection of surface stresses with velocity field is the stipulation that these stresses are proportional to strains. The constant of proportionality is known as the viscosity.

The partial differential the Navier-Stokes equations for 3D case can be presented as follows:

$$\begin{aligned}
\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)&=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right),\\
\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)&=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right),\\
\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)&=\rho g_z-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right).
\end{aligned} \tag{2}$$

Computer simulations were conducted considering the following assumptions that are adequate to the given system: incompressible fluid, three-dimensional flow, the flow is laminar and the mass force is the gravity force. Additionally, the model implements the following equilibrium equations to solve the flow problem:

- conservation law of mass:

$$\frac{\partial\rho}{\partial t}+\nabla\left(\rho u\right)=0, \tag{3}$$

- conservation law of momentum

$$\rho\frac{\partial u}{\partial t}=\nabla\sigma+\rho f, \tag{4}$$

- conservation law of energy

$$\rho\frac{d}{dt}\left\{TC_p+\frac{u^2}{2}\right\}=\rho fu+\rho q+\nabla(\Gamma\operatorname{grad}T)+\nabla(\sigma u), \tag{5}$$

where:
σ - stress tensor,
T - temperature,
Γ - thermal conductivity,
C_p - volumetric heat capacity,
q - capacity of internal heat source.

3. Numerical solution description

In order to solve (with the use of 3D) the equations presented in chapter 2, the researcher used the models that can imitate flow area real shapes. The numerical solution implements eight-node solid finite elements. The finite element used allowed conducting of the Navier-Stokes equations integration process in Euler coordinate system. Navier-Stokes equations integration process used Gaussian quadrature. Introduction of one integration point was supposed to shorten calculation time. Integration of dynamic equation was conducted implementing explicit scheme.

4. Simulation of blood flow in a syringe

In order to calculate the blood flow in a syringe the researcher used a model that allows adhesive fluid flow. The model uses Navier-Stokes equations, which were solved in Euler domain. In this system adhesive fluids move against a stationary mesh that copies a flow canal outline. By means of the mesh it was possible to imitate the surface that limits the fluid flow area inside the syringe- test tube system. The system was simplified i.e. the details relating to the front side of the test tube and the internal needle canal were not included as presented in Fig. 1.

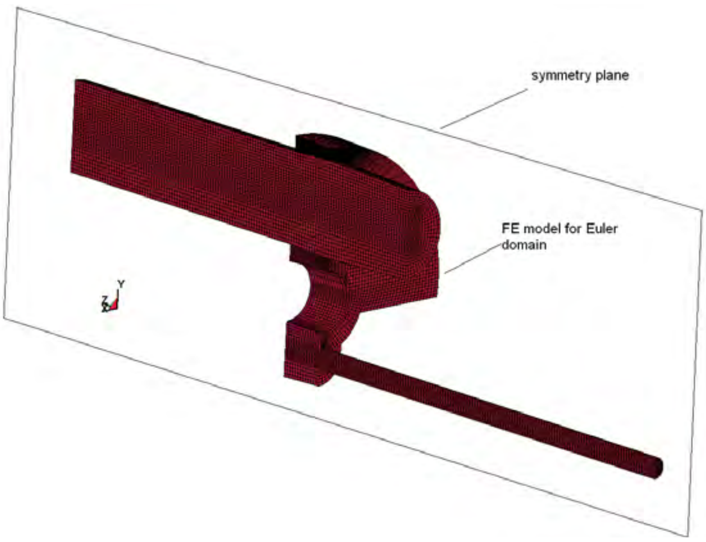

Fig. 1. FE model and symmetry conditions

The researchers precisely imitated the shape of flow canal between the anti-reflux valve and the front side of the set handle. In order to reduce the numerical task size the researchers used syringe - test tube flow canal symmetry. This allowed using a half of the flow canal model, in which appropriate boundary conditions resulting from the symmetry used were defined.

5. FE model of blood flow in the syringe

Numerical model of the flow area in the syringe was built using the flow in relation to symmetry surface as presented in Fig. 1. The considered model version did not include the influence of a flow canal surface limitation. In fact, the flow canal is limited to internal needle canal. This case was further considered in next versions of the numerical analysis presented in this paper. On the basis of the assumptions used while developing FE model [5], the researchers introduced the boundary conditions discussed in detail in Fig. 1.

Boundary condition No. 1 (Fig. 2) was defined in the nodes placed on the external surface and it imitates fluid flow area partition. Boundary condition No. 2 was defined in the nodes placed on symmetry surface. Appropriate pressure conditions were introduced to the area represented by the nodes placed on the inlet and outlet of the flow area. Tab. 1 presents statistics of the FE model described in Euler coordinate system. The diameter of the inlet and outlet area was respectively 24.8 and 5.37 mm.

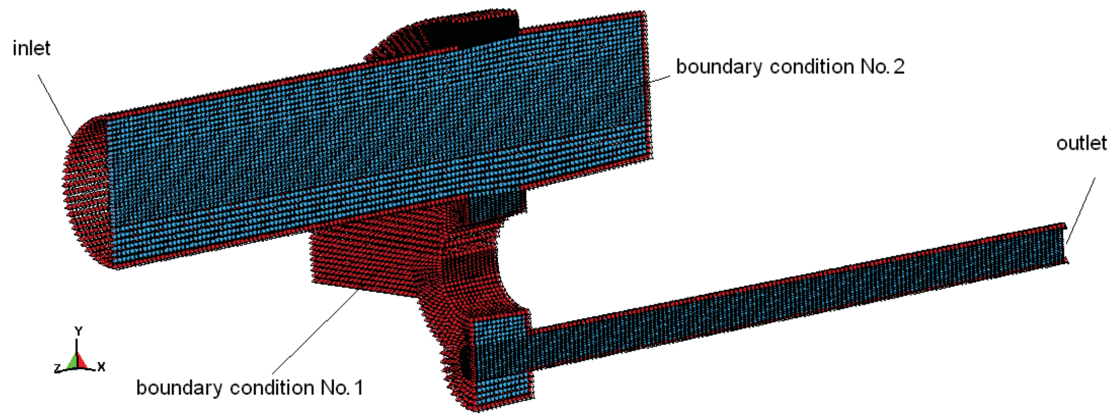

Fig. 2. Boundary conditions defined in FE model

Tab. 1. FE model statistics

	Number of nodes	Number of elements
Flow volume	88247	78463

Blood parameters such as viscosity and density were included in the computations. Viscosity (internal friction) is a feature of fluids and plastic solid substances that characterizes their internal friction against the flow. Friction against flow that appears on the border of fluid and vessel side is not viscosity. Viscosity is one of the most important fluid (liquid and gas) features. In this problem the blood viscosity equals μ = 20·10^{-4} [kg/m·s] was implemented. The blood is 2.5 times more sticky in temperature 0ºC than in 37ºC. Blood viscosity depends on hematocrit value (a ratio of blood cell volume to blood volume), temperature (exponential viscosity nature vs. temperature) and vessel cross-section.

Another feature that is characteristic for fluids is their density. Homogeneous fluid density is defined as a fluid mass divided by the volume (in kg/m^3 or g/cm^3). Fluid density may depend on several factors such as temperature or pressure. Fluid density changes slightly even with significant temperature or pressure differences. Thus, we call them incompressible fluids. In this problem the blood density equals ρ = 1.06·10^3 [kg/m^3] was assumed.

6. Numerical analysis – pressure change on the inlet and outlet

Numerical analysis was conducted with the use of CFD (Computational Fluid Dynamics) module included in LS-DYNA system [5]. The load in each version constituted pressure held to inlet front side of limitation area of the flow canal filled with solid model of fluid domain. This version used pressure load held to inlet area (Fig. 3) in the form of a variable function. The base value implemented for calculations was 101325 Pa (Fig. 3).

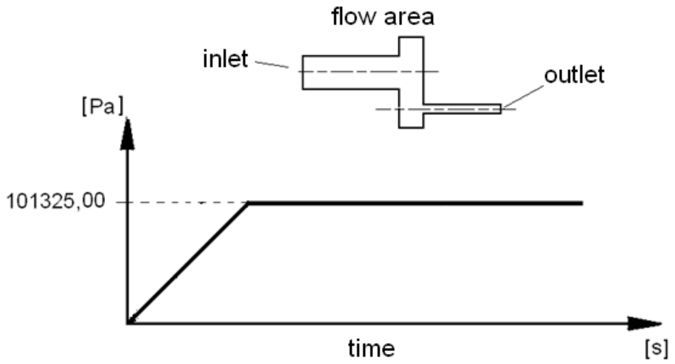

Fig. 3. Pressure load adopted on the inlet and outlet of the flow area

Figure 4 presents velocity isosurfaces along *Z* axis i.e. along the flow area. The isosurfaces were created from current line for the pressure 10% higher than the base one. Fig. 4 proves that the most significant velocity changes appear in the cross-section change area in the flow area i.e. while the blood flows into outlet ended pipe. This gives evidence that the blood is mixed.

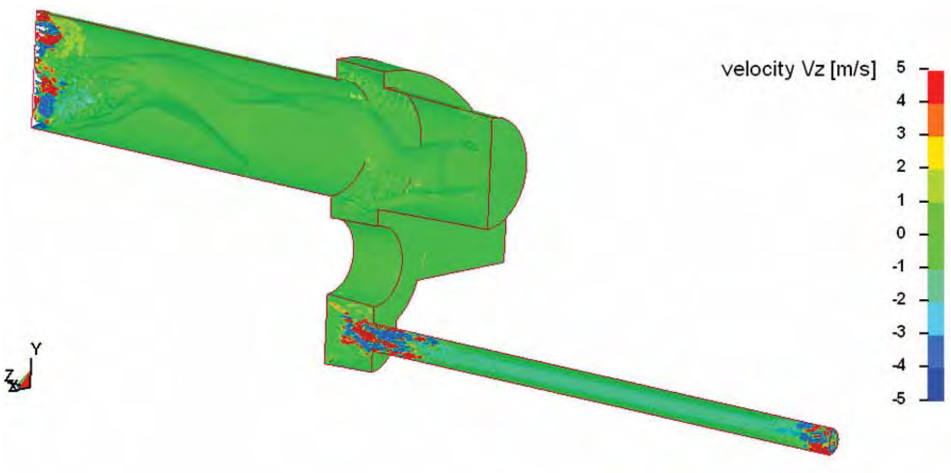

Fig. 4. Streams while flowing along Z axis

In addition, Fig. 5 presents isosurfaces with velocity vector value. Velocities are calculated on the basis of the vector constituents on *X*, *Y* and *Z* axis. Fig. 6 presents flow velocities routes in sample points (nodes) „*A*" and „*B*". In point "*A*" flow velocity changes in the range of 0.1 to – 0.5 m/s, whereas in point "*B*" flow velocity achieves maximum value equal -7.8 m/s after 0.06 s. The flow stabilizes after about 0.5 s. Velocity of stabilized flow in the area of external needle outlet canal is determined on the level of -3 m/s. Symbol "-" signifies reverse turn of velocity vector with reference to *Z* axis turn.

7. Numerical analysis – constant pressure on the inlet for flow with an orifice version

The researchers also considered flow analysis versions which in an outline of the flow canal (constituting fluid move limitation area) included flow choking. The flow choking was the result of flow area narrowing to internal needle flow canal. The needle unclogs the fluid outflow from the test tube to the canal between front sides of anti-reflux valve and the set handle. It was included in a discrete model by placing a half of front side of choking side with an orifice. It corresponds with the real internal needle canal orifice through which the fluid flows.

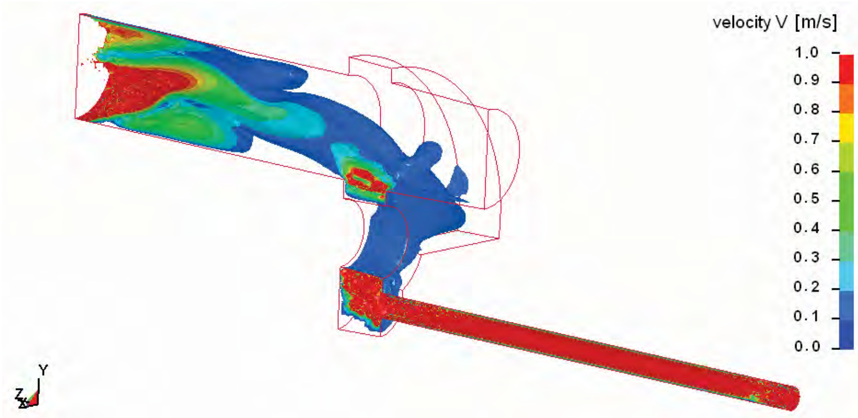

Fig. 5. Streams while flowing in a syringe

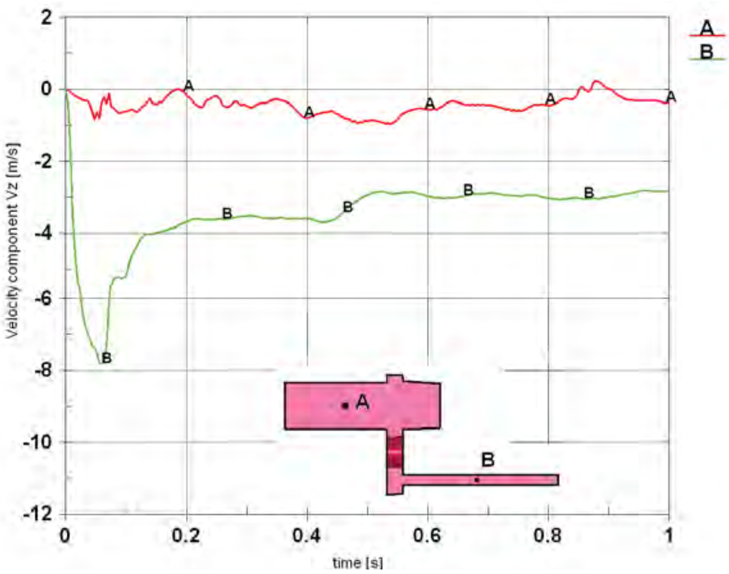

Fig. 6. Flow velocity routes on Z axis in points A and B

This analysis version used pressure load in the form of a variable function. Pressure load was defined on the inlet area (Fig. 7). The base value implemented for calculations was 101325 Pa. The value equals zero was defined on the outlet. Moreover, Figs. 8 and 9 present isosurfaces with velocity vector values. Velocities are calculated on the basis of the vector components on *X*, *Y* and *Z* axis.

Figure 10 presents flow velocity routes in sample points (nodes) "*A*" and "*B*". In point "*A*" flow velocity achieves maximum value equal -0.6 m/s in initial flow phase. In further phases it stabilizes at -0.2 m/s. In point "*B*" the flow stabilizes after a sudden velocity increase at the moment of extortion effect at -3.0 m/s. The flow is determined after about 0.5 s. Symbol "-" signifies reverse turn of velocity vector with reference to *Z* axis turn.

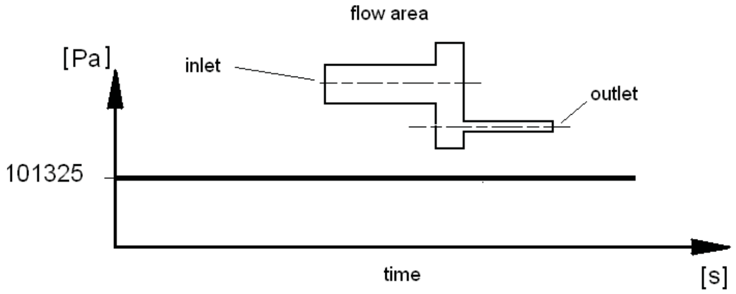

Fig. 7. Pressure load adopted on the flow area

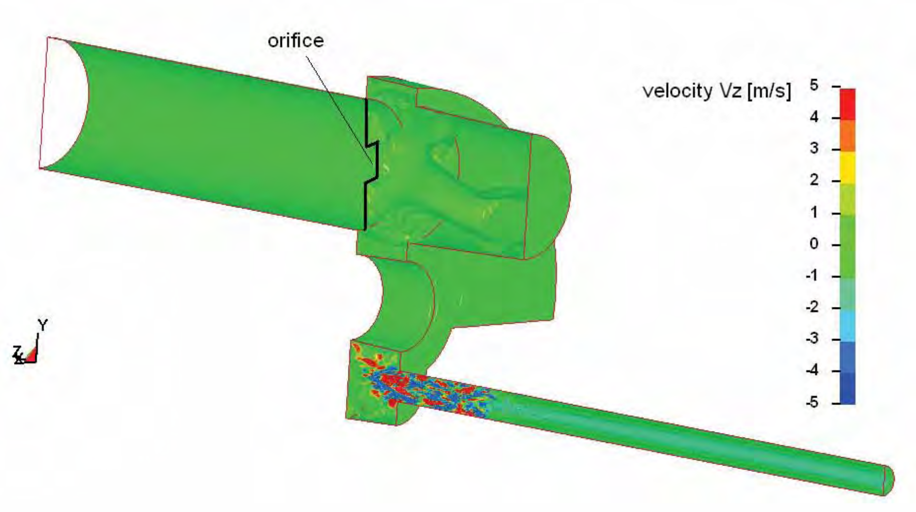

Fig. 8. Streams while flowing along Z axis for syringe with an orifice

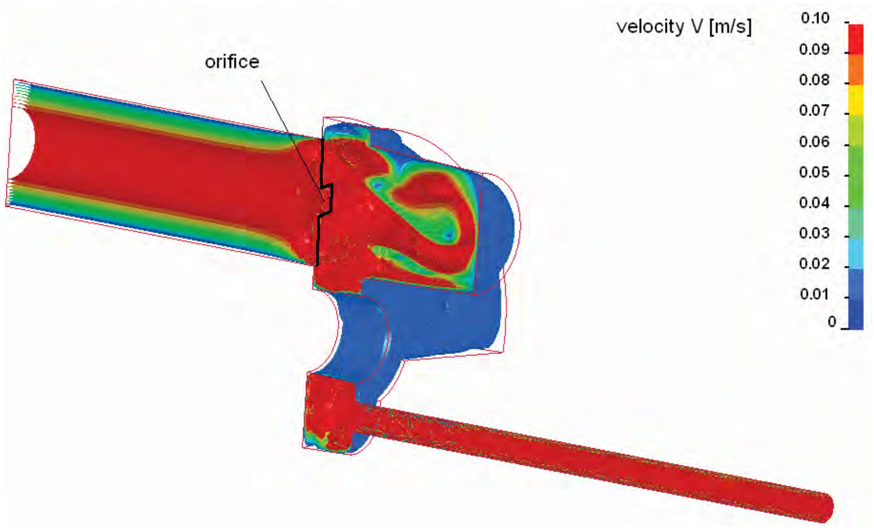

Fig. 9. Streams while flowing in a syringe with an orifice

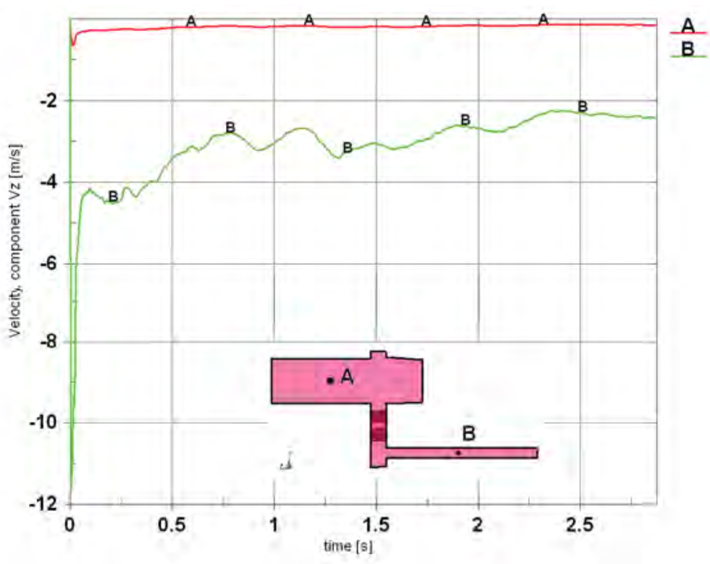

Fig. 10. Flow velocity routes on Z axis in points A and B for flow with an orifice

8. Conclusions

While considering the problem, the researchers analyzed blood flows through the area that borders on the sides of the piston, syringe and needle sides. Pressure arises after the initiation of the piston's move and the blood flows as a result of the difference in pressures. The blood flows from the areas with higher pressure to the ones with lower pressure. The numerical research focused on analysis of the boundary conditions influence on the arrangement of the velocity and blood pressure in a syringe. In order to calculate the blood flow through a syringe the researchers used a model that allows the flow of adhesive fluids - substances with internal friction. Navier-Stokes' equations were used to describe adhesive fluid move mathematically. These equations allowed describing non-stationary flow of incompressible fluids, i.e. including no volumetric changes, however, including figural changes. On the basis of the analysis conducted it is possible to draw the following conclusions:

- Pressure conditions increase procedure application both on the inlet and on the outlet provides correct results. Fluid velocity value in the outlet area decreases to about 3 m/s.
- Difference in pressures appears on the inlet to the needle in each of the analyzed cases. In this place, i.e. in front of the inlet to the needle, we observe vorticity, i.e. mixing of fluids before they flow into the needle. This phenomenon is the most appropriate from practical point of view.

In last analyzed case, i.e. with flow orifice, fluids dam up in front of the orifice and then flow into the next chamber just after the orifice. The fluids flow to the chamber as a piled up stream. Then, the fluids hit internal bend sides and rotate again in front of the needle orifice entry. Considering fluids parameters, vorticity process improves fluid mixing process. At the same time, fluid flow velocity decreases in the needle to about 2.5 m/s.

References

[1] Munson, B. R., *Fundamentals of fluid mechanics*, John Wiley & Sons, 1994.
[2] Fortuna, Z., Macugow, B., Wąsowski, J., *Metody numeryczne*, WNT, Warszawa 2005.
[3] Gryboś, R., *Podstawy mechaniki płynów*, PWN, Vol. I & II, Warszawa 1998.
[4] Tuliszka, E., *Mechanika płynów*, PWN, Warszawa 1980.
[5] Hallquist, J.O., *LS-Dyna. Theoretical Manual*, California Livermore Software Technology Corporation, 2005.