Scientific quarterly journal ISNN 1429-7264

Agricultural Engineering

2014:1(149):175-184

Homepage: http://ir.ptir.org

DOI: http://dx.medra.org/10.14654/ir.2014.149.019

SIMULATION TESTS OF FLUID FLOW IN THE PIPELINE ELEMENTS[1]

Joanna Piepiórka-Stepuk*, Sylwia Mierzejewska

Department of Food Industry Processes and Facilities, Koszalin University of Technology

*Contact details: ul. Racławicka 15-17, 75-620 Koszalin, e-mail: joanna.piepiorka@tu.koszalin.pl

ARTICLE INFO

Article history:
Received: December 2013
Received in the revised form: January 2014
Accepted: February 2014

Keywords:
pipelines
cleaning in the CIP system
CFD calculations

ABSTRACT

The paper presents research results concerning the use of commercial software used for calculations in the fluid mechanics. With the use of numerical methods of CFD in the selected elements of pipe installations, pressure, speed and shear stress distribution on their walls were presented and analyzed in the aspect of cleaning conditions in the CIP system. The tests which were carried out constitute a part of the tests concerning conditions of cleaning installations of production installations funded with the research subsidy. The obtained research results have an interdisciplinary character whereas their interpretation with reference to the cleaning conditions confirms rightness of using the CFD method for forecasting and hygienic modeling of food industry devices.

Introduction

Cleaning in the CIP system (Clean In Place) is an automatized process, which does not require dismounting cleaned elements and consists in flowing through the cleaned installation suitable cleaning solutions, which during the flow moisten and tearing off post-production sediments. Devices and all production lines, which take part in production of liquid and semi-solid food along with pipelines for transportation of raw material in order to carry out another technological operation, are subject to cleaning in the CIP system (Diakun, 2013). The structure of pipelines can cause secondary infection of the produced food, particularly when it is equipped with many elbows, tees, dead ends and extra valves (Lelievre et al., 2002b, Jensen et al., 2005; Mierzejewska et al., 2013). These are the elements with variable cross-sections of flow channels, which affect flow conditions. These, on the other hand, are the most important factor in the process of cleaning within the flow, which is crucial for obtaining clean surfaces. The hydro-mechanical fluid impact on walls during the cleaning process provides mechanical cleaning effect and favors breaking off sediment particles from the cleaned surfaces, their dispersion in the entire fluid volume, their transport and

[1]Research financed by the the Polish Ministry of Science and Higher Education Research under project No. N N313 136838

removal. It was also shown that these impacts affect the reduction of the amount of microorganisms as early as during the cleaning process (Grasshoff, 1992; Lelievre et al., 2002a) and that the detachment of bacterial cell from the surface which isbeing cleaned, occurs when the wall shear stresses are greater than adhesive forces responsible for the bacterial adhesion (Hermanowicz et al., 1989; de Jong et al., 2002).). Many authors show, that the kinetic removal of contaminations is a function of fluid flow cleaning solutions, Reynolds number (1) and shear stresses forming on the wall (2) (Lelievre et al., 2002a; Jensen et al., 2005; Blel et al., 2007; Diakun et al., 2010), where wall shear stresses τ_w are defined as a product of dynamic viscosity μ and the flow velocity at a certain distance from the wall of element y (2).

$$Re = \frac{\rho \cdot u \cdot d}{\mu} \tag{1}$$

where:
Re – Reynolds number,
ρ – density, ($kg \cdot s^{-3}$)
u – fluid flow, ($m \cdot s^{-1}$)
d – characteristic dimension (diameter), (m)
μ – dynamic viscosity coefficient, ($kg \cdot m^{-1} \cdot s^{-1}$)

$$\tau_w = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0} \tag{2}$$

where:
τ_w – shear stresses, (Pa)
μ – dynamic viscosity coefficient, ($kg \cdot m^{-1} \cdot s^{-1}$)
u – fluid flow, ($m \cdot s^{-1}$)
y – fluid distance from the wall of the element, (m)

Velocity and flow turbulence (Re>10000) and complete moistening of all surfaces which are in contact with the product have a huge impact on the conditions of deposits removal and they are basic prerequisite in obtaining clean surface of pipelines (Bansal & Chen, 2005; Piepiórka & Mierzejewska, 2009). In order to determine the flow conditions occurring in some parts of pipelines, the numerical calculations with the CFD code were used (Lelievre et al., 2002a; Lelievre et al., 2002b; Jensen et al., 2005; Rahaman et al., 2007). It is a tool widely used for modeling and predicts flow conditions in the closed industrial installations, mainly when it is difficult to carry it out in laboratory conditions.

The objective, material and methods of the research

The objective of the research was to:

– Determine the conditions of fluid flow in the selected elements of pipelines: elbows, T-connectors, dead ends, using numerical calculations basing on the CFD code;

– Indicate, on the basis of numerical tests, in the above mentioned elements, the areas which are hard to clean within the flow.

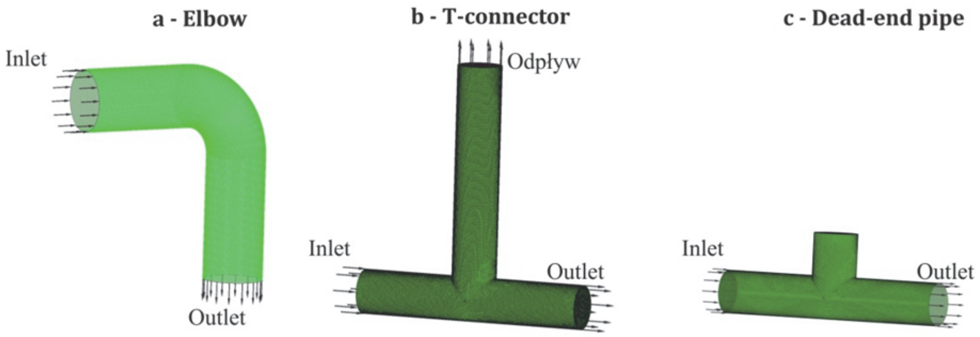

Figure 1. Boundary conditions for flow a – in the pipeline elbow; b – in the T-connector, c – in dead ends

$$u_x = 0; u_y = 0; u_z = 0; \tag{3}$$

$$\dot{m}_{in} \geq 0; \dot{m}_{out} \geq 0; \tag{4}$$

A half-empirical model of turbulence $\kappa - \varepsilon$ was used for analysis of the fluid flow in a pipeline. This model is properly adapted for the flows with high turbulence. Thus, the turbulent kinetic energy is cascade-transferred from large scale whirls to the small scale ones, where it is dissipated. In the concept of this model, the system of equations (N-S) is closed with two additional differential equations: the kinetic energy transport "κ" and the dissipation rate of turbulent kinetic energy "ε" (Elsner, 1987; Kazimierski, 2004; Bogusławski et al., 2008).

The equation for kinetic energy κ

$$\rho u_x \frac{\partial \kappa}{\partial x} + \rho u_y \frac{\partial \kappa}{\partial y} + \rho u_z \frac{\partial \kappa}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_e}{\sigma_\kappa}\frac{\partial_\kappa}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_e}{\sigma_\kappa}\frac{\partial_\kappa}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_e}{\sigma_\kappa}\frac{\partial_\kappa}{\partial z}\right) + G - \rho\varepsilon \tag{5}$$

The equation for the dissipation rate ε

$$\rho u_x \frac{\partial \varepsilon}{\partial x} + \rho u_y \frac{\partial \varepsilon}{\partial y} + \rho u_z \frac{\partial \varepsilon}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_e}{\sigma_\varepsilon}\frac{\partial_\varepsilon}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_e}{\sigma_\varepsilon}\frac{\partial_\varepsilon}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_e}{\sigma_\varepsilon}\frac{\partial_\varepsilon}{\partial z}\right) + C_1\frac{\varepsilon}{\kappa}G - C_2\rho\frac{\varepsilon^2}{\kappa} \tag{6}$$

where:
κ – turbulence kinetic energy, ($m^2 \cdot s^{-2}$)
ε – turbulence dissipation rate, ($m^2 \cdot s^{-3}$)
G – generation of the turbulence kinetic energy
μ_e – turbulent viscosity, (Pa·s)

Equations in the adopted turbulence model $\kappa - \varepsilon$, show the evolution of parameters which determine the turbulence (Jones & Launder, 1972; Wilcox, 1998). It was assumed that the constant of the model for turbulence is $C\mu=0.09$ while constant values of the coefficients are $\sigma_\kappa=1.0$; $\sigma_\varepsilon=1.3$; $C_1=1.44$; $C_2=1.92$.

The numeric calculations have been carried out by means of the Finite Volume Method (FVM), with the use of Ansys CFX 12 software. The idea of this method is to split computational domain into small cells – the finite volumes – and to enforce conservation by prescribing fluxes at the cells interfaces. In this way the evolution of the conserved quantities can be approximated if the fluxes are suitable approximations of the fluxes given by the conservation laws. From the pipeline elements with a diameter of d=0.038 m, spaces, which were forming the flow channels, were isolated. A numerical grid with the size of an element of 0.001m was used for discretization of each model. For this size of cells, there was no significant effect on the final simulation result. The numerical grid had tetrahedral-shaped cells, which constituted areas of computation. Boundary and initial conditions at the first step of the first iteration, were defined as a zero velocity value on the walls of the tested models, zero values of turbulent kinetic energy coefficients κ and the dissipation of turbulence ε, mass flow rate at the inlet and the outlet of the tested models at $\dot{m}=1.95$ kg·s^{-1} level. Surface roughness was assumed at the level of $R_a=0.4$ µm. The adopted flow rate was determined on the basis of experimental measurements of the flow velocity in the real channel pipe, for which the average flow velocity was $u=1.5$ m·s^{-1}. The flow was determined as turbulent, while water was the flowing liquid with viscosity and density corresponding to water at the temperature of T=45ºC. The flow conditions were established as isothermal and the flow was described as stable. On the basis of the maps of flow velocity, streamlines, velocity vectors and pressure distribution areas were determined where the flow may be advantageous or disadvantageous for the cleaning processes in the flow of t-connectors, elbows and dead ends in pipelines.

Research results

For numerical calculations the geometric models were developed (fig. 1) and the flow conditions in the selected elements of pipelines were modeled. The results of the research were shown in figures below. The first part presents the results of numerical tests for piping elbows, T-connectors and dead ends (fig. 2, 3, 4, 5). Then, in the above elements the areas which could be difficult to clean in the standard CIP method were presented (fig. 6).

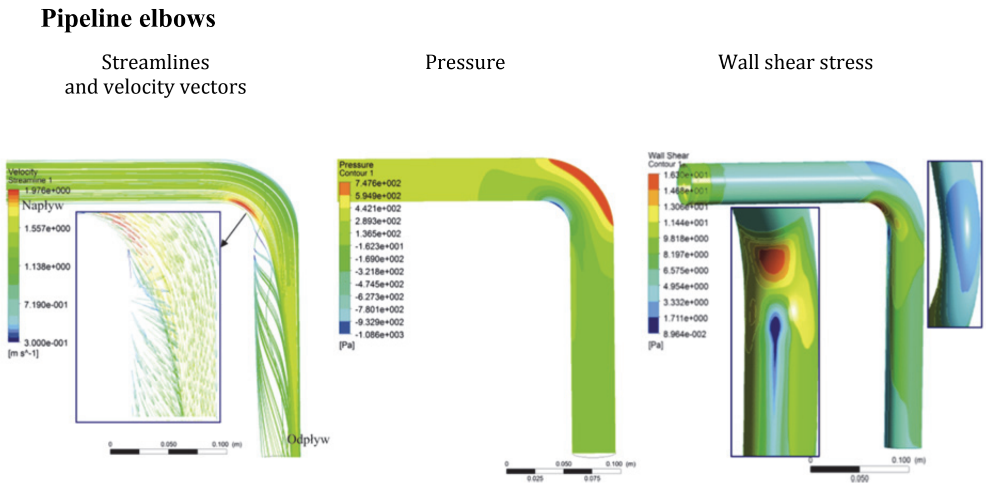

Figure 2. Results of numerical tests for pipeline elbows

The conducted numerical analyses indicate that the largest flow velocity and a large velocity gradient are present at the internal radius of the elbow. The lowest flows occur below the curve and on the external radius. In these areas small velocity gradients are present, too. The analysis of velocity vectors and streamlines leads to the conclusion that the most unfavorable area is the pipe surface below the internal curve. In these areas separation of fluid flow, as well as changes in the flow direction, occur. Whirls of fluid flow are formed here which is disadvantageous for the cleaning processes. The flow velocities in these places dropped to the value close to 0 m·s^{-1} and the wall shear stress dropped as well to the value of ≈0.009 Pa. The pressure distribution in the tested model indicates that the greatest pressure values occur on the outer edge of the elbow and decrease with the reduction of distance to the inner edge of the elbow. Pressures should be read as gradients for the reference pressure of P_r=1 atm.

Subsequent studies were carried out for elbows with a diameter of d=0.051 m and d=0.076 m. The numerical results were presented in figure 3.

The results of numerical calculations of fluid flow, as a distribution of velocity fields in tested models illustrate that the greatest velocities occur on the internal edges of the pipeline elbow. With the assumed values of the flow rate of fluid you can see a change of velocity due to the increased diameter. For pipeline elbow with a diameter of d=0.038 m the maximum flow velocity which occurs on the outer edge obtains u=1.97 m·s^{-1} while for diameter d=0.051 m it obtained u=1.15 m·s^{-1} and for diameter d=0.076 m it obtained only u=0.54 m·s^{-1}.

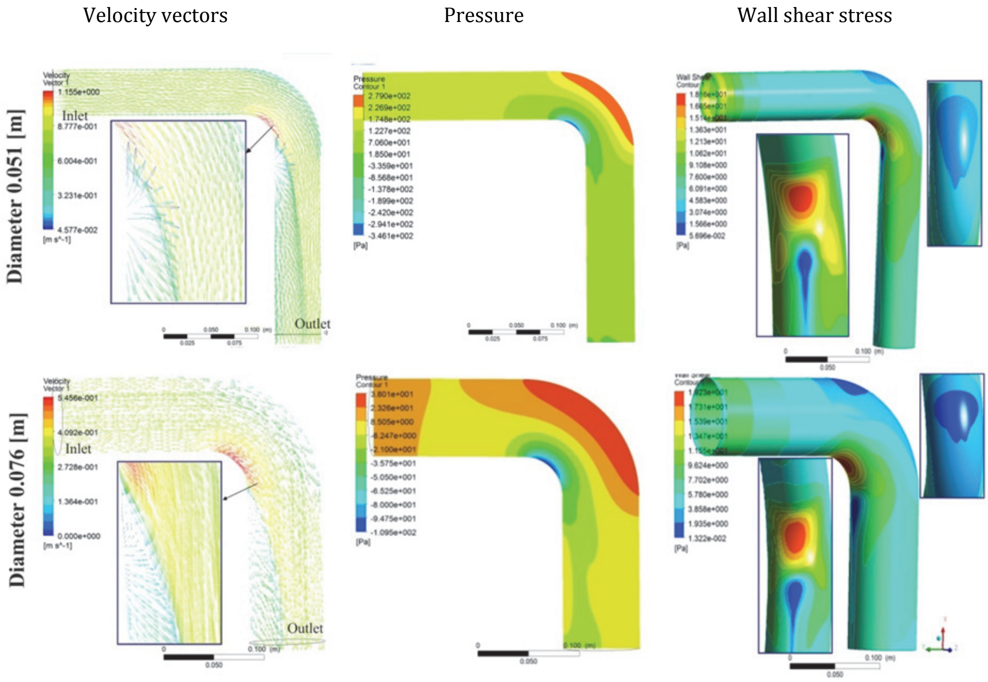

Figure 3. Results of numerical tests for pipeline elbows of various diameter

For the same models the numerical simulations with higher mass flow rate (m=3 kg·s^{-1} m=6 kg·s^{-1}) were made. The results, however, did not bring anything new. There were no significant changes in the flow character in the tested models except for the resulting flow velocity values. Relatively higher values of mass flow achieved a higher flow velocity - for smaller ones, lower values. Therefore, further studies have not taken into account this factor and focus only on the geometry of the tested elements.

Based on the results of numerical analyses, two areas with difficult cleaning conditions were identified in the elbow; these are the areas below the internal arc of the elbow and the area on the external arc (fig. 6).

T-connectors

The results of numerical calculations for tees show that the largest flow rate is obtained on the external wall, located perpendicularly to the fluid flow direction. A sudden change in the flow direction of the flowing fluid causes adverse whirls on the wall of the T-connector at the inflow. The reduced values of the flow velocity in these areas (u≈0.4 m·s^{-1}), could result in forming and growing the sediment layers and disadvantageous cleaning conditions. The lower flow velocities occur also on the outlet of the analyzed T-connector. From the initial value of u=1.5 m·s^{-1}, the flow velocity drops to u=0.1÷0.3 m·s^{-1}. This is due to liquid separation. There are no whirls of liquid in these areas, but the basic requirement for cleaning in place, which is a turbulent flow, is fulfilled only in the minimal degree. For the flow velocity at the outlet of the Reynolds number is obtained and it is $Re \approx 6000 \div 19\,000$ and therefore the flow has a laminar character. For this reason the wall shears stress are lowered.

The highest values of wall shear stresses for a tee were achieved in areas where the greatest values of flow velocity occur. That is, on the right wall of branching. On this basis, three areas in tees which could be difficult to be cleaned during cleaning in the flow were determined. That is the left wall of tees branching and the upper surface of the pipe per branching (fig. 6).

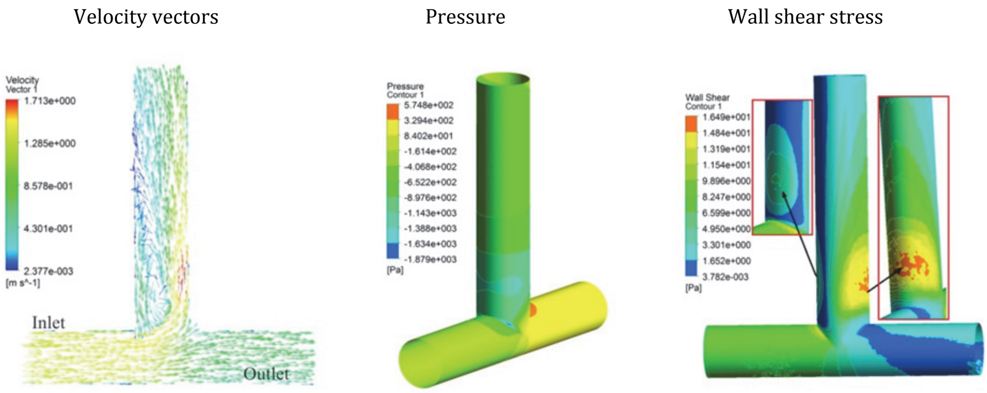

Figure 4. Distribution of flow tracks and flow velocity, pressures and shear stress vectors in a T-connector

Dead ends

In case of dead ends, liquid whirls occur as a result of the inflow in empty zones. Numerical analyses show that in long dead pipelines stasis can be also observed. The decrease in the media flow velocity in the pockets causes both product and cleaning media retention, particularly if they are directed downwards. Therefore, such solutions should be particularly avoided. However, in tees directed upwards one should expect difficult access for cleaning agents. The sediments which accumulated there will not be cleaned and these will be a great area for growth of microflora. The presence of a dead end also causes a 50% decrease in the flow velocity, to the value of u=0.5÷0.7 m·s^{-1} in the extension of the pipeline (right tube), on its upper edge. This area also cannot be cleaned properly.

The highest values of wall shear stresses in dead ends are at the inflow of liquid. Numerical test results also show that it is more preferable to use short blind ends. The areas considered as a potential hazard in the cleaning process were shown in figure 6.

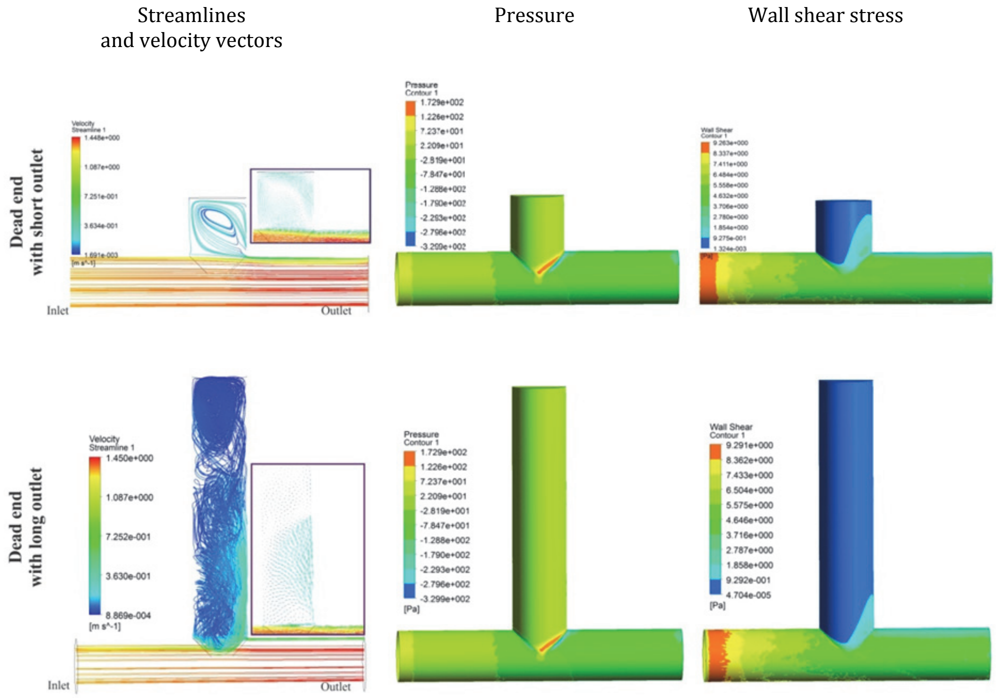

Figure 5. Distribution of flow tracks and flow velocity, pressures and shear stress vectors in dead ends of pipelines

On the basis of the CFD calculation, based on hydro-mechanical flow interactions on the walls of an item being cleaned, it was possible to identify zones in which favorable and unfavorable conditions for cleaning may occur. The identified areas were presented in figure 6.

The areas in pipeline elements which are identified as difficult to be cleaned in the flow

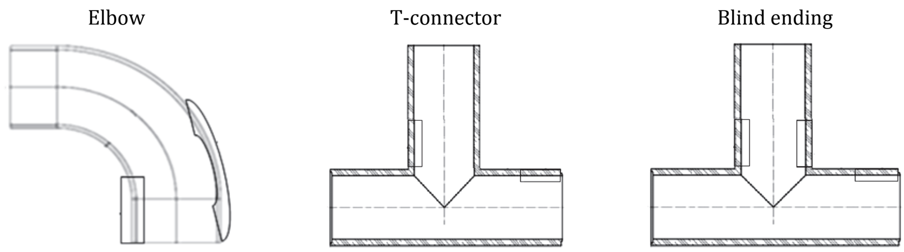

Figure 6. Areas in the tested elements of pipelines exposed to insufficient cleaning

Conclusion

The nature of the flow channel of a given device or installation has a significant impact on the hydrodynamic conditions of the liquid flow and, thus, on the conditions of the CIP cleaning. Depending on the shape of the channel, the variable velocity profiles and distribution of pressure and consequently of local variable shear stresses were observed. With respect to the cleaning conditions this may affect the varying degree of cleanliness in different areas of the tested elements in the standard procedure of the CIP.

The liquid flow in industrial transporting systems in the closed arrangement prevents finding the areas in pipes which may suffer from insufficient cleaning. This was, however, possible by conducting simulated computer calculations using different computer methods. The resulting velocity distributions indicate that there are significant differences of the flow velocity values in the analyzed models as well as differences in the flow velocity in the specified areas of piping elements. Taking into account the initial value of the fluid flow, the differences in the flow rates reach 0.7 $m \cdot s^{-1}$ in the case of elbows, or even close to 1.5 $m \cdot s^{-1}$, in case of blind ends. On the basis of the presented results it was shown that the areas of elbows and T-connectors are the most difficult to clean on internal curves. On the other hand blind ends are pockets where stagnant liquid may occur and if there is a need to use them, short blind ends should be used if possible. Furthermore, the amount of this type of elements in transporting pipeline installations should be minimal.

The CFD calculations are an innovative tool used for solving engineering problems. The research allowed obtaining information on the fluid flow velocity distributions in the pipe elements with different shapes. The velocity distributions, pressure and wall shear stress, show the changing nature of the flow in these elements, as a result of which in flow channels uneven distribution of liquid occurs.

References

Bansal, B.; Chen, X.D. (2005). Fouling of heat exchangers by dairy fluids – a review. *Heat Exchanger Fouling and Cleaning - Challenges and Opportunities* RP2 (23).

Blel, W.; Benezech, T.; Legentilhomme, P.; Legrand, J.; Le Gentil-Lelievre, C. (2007). Effect of flow arrangement on the removal of *Bacillus* spores from stainless steel equipment surfaces during a Cleaning In Place procedure. *Chemical Engineering Science, 62,* 3798-3808.

Bogusławski, A.; Drobniak, S.; Tyliszczak, A., (2008). Turbulencja – od losowości do determinizmu. *Modelowanie Inżynierskie,* (5)T.36, 41-48.

de Jong, P.; Te Giffel, M. C.; Straatsma, H.; Vissers, M. M. M. (2002). Reduction of fouling and contamination by predictive kinetic models. *International Dairy Journal, 12,* 285-292.

Diakun, J. (2013) Review, scheme and cleaning methods analysis. *Polish Journal of Food Engineering* 1/4-2013(5), 5-10. http://ips.wm.tu.koszalin.pl/

Diakun, J.; Piepiórka, J.; Bil, T. (2010). Polioptymalizacja warunków mycia rurociągów. *Postęp w Inżynierii Żywności, 546*, 69-76.

Elsner, J.W., (1987). *Turbulencja przepływów.* Warszawa, Wydawnictwo PWN, ISBN: 83-01-0613-4

Grasshoff, A., (1992). Hygienic design: the basis for computer controlled automation. *Food and Bioproducts Processing. Transactions of the Institution of Chemical Engineers,* C 70, 69-77.

Hermanowicz, S.W., Danielson, R.E., Cooper, R.C. (1989). Bacterial deposition on and detachment from surfaces in turbulent flow. *Biotechnology and Bioengineering, 33*, 157-163.

Jensen, B.B.B; Friis, A.; Bénézech, T.; Legentilhomme, P.; Leličvre, C. (2005). Local wall shear stress variations predicted by computationalfluid dynamics for hygienic design. *Food and Bioproducts Processing.Transactions of the Institution of Chemical Engineers,* C 83(1), 1-8, DOI.10.1205/fbp.04021.
Jones, W. P.; Launder, B. E. (1972), The prediction of laminarization with a two-equation model of turbulence. *International Journal of Heat and Mass Transfer, 15,* 301-314.
Kazimierski, Z., (2004). *Podstawy mechaniki płynów i metod komputerowej symulacji przepływów*. Łódź, Politechnika Łódzka, ISBN: 8372831335.
Lelievre, C.; Legentilhomme, P.; Gaucher, C.; Legrand, J.; Faille, C.; Bénézech, T. (2002a). Cleaning in place: effect of local wall shear stress variation on bacterial removal from stainless steel equipment. *Chemical Engineering Science, 57*(8), 1287-1297, DOI. 10.1016/S0009-2509(02)00019-2.
Lelievre, C.; Antonini, G.; Faille, C.; Bénézech, T. (2002b). Cleaning-in-place, modeling of cleaning kinetics of pipes soiled by Bacillus spores assuming a process combining removal and deposition. *Food and Bioproducts Processing, Transactions of The Institution of Chemical Engineers,* C 80(4), 305-311, DOI.10.1205/096030802321154826
Mierzejewska, S.; Diakun, J.; Piepiórka-Stepuk J.; Zając M. (2013). Skuteczność mycia w systemie CIP wybranych elementów instalacji przesyłowych. *Nauki Inżynierskie i Technologie, 3*, 70-79.
Piepiórka, J.; Mierzejewska, S. (2009). Instalacje rurowe – mycie w obiegu zamkniętym. *Przemysł Spożywczy, 11*, 29-32.
Rahaman, M.F.; Bari, S.; Veale, D. (2008). Flow investigation of the product fill valve of filling machine for packaging liquid products. *Journal of Food Engineering, 85*, 252-258.
Wilcox, D. C. (1998). *Turbulence Modeling for CFD*. Second edition. Anaheim: DCW Industries

BADANIA SYMULACYJNE PRZEPŁYWU CIECZY W ELEMENTACH RUROCIĄGÓW

Streszczenie. W pracy przedstawiono wyniki badań dotyczące wykorzystania komercyjnych aplikacji komputerowych stosowanych do obliczeń w mechanice płynów. Za pomocą numerycznych metod CFD, w wybranych elementach instalacji rurowych, przedstawiono rozkłady ciśnienia, prędkości i naprężeń ścinających na ich ścianach i poddano je analizie w aspekcie warunków mycia w systemie CIP (czyszczenie na miejscu). Przeprowadzone badania stanowią cześć badań dotyczących warunków mycia instalacji produkcyjnych finansowanych w ramach grantu badawczego. Uzyskane wyniki badań mają charakter interdyscyplinarny, natomiast ich interpretacja w odniesieniu do warunków mycia potwierdza słuszność stosowania metod CFD do prognozowania i higienicznego modelowania urządzeń przemysłu spożywczego.

Słowa kluczowe: rurociągi, mycie w systemie CIP, obliczenia CFD