Numerical Simulation of Separated currents and Heat Transfer in a Channel with Discrete Roughness

Natalia Sidenko, Egils Dzelzitis

Riga Technical University
nsidenko@inbox.lv, egils.dzelzitis@rtu.lv

INTRODUCTION

Studies of heat transfer and hydrodynamics of separated currents have great practical importance. Such currents occur during flowing around depressions of various shapes and those are one of the most relevant and interesting areas of fluid mechanics.

Usage of heat exchange surfaces with depressions gives energy advantage. Often the increase in heat transfer caused by the usage of depressions is goes with a slight increase in hydraulic resistance. For example, in studies of flow in channels with intensifiers in the form of spherical depressions, which were made in works [1-3], it has been experimentally established that heat exchange surfaces with semi spherical depressions allow a significant increase (by 1.5-4.5 times) in heat exchange at a moderate growth in the hydraulic resistance.

Inside and near spherical depressions occur large-scale dynamic vortex structures, which can be observed in a wide range of flow regimes. A series of contributions exists, in which are reviewed numerous aspects of vortex formation in spherical depressions, when impacted by the wide range of factors [4-8].

Although the large number of studies devoted to the review of problems of surfaces with spherical depressions has been published, several questions remained unanswered.

This caused by the complexity of the phenomenon, which is being impacted by velocity and the defining temperature of the flow; the latter in turn determine the values of the selected physical parameters.

Basic methodology of numerical simulation of hydrodynamics and heat transfer in a channel with discrete roughness, was reviewed in publication [9].

The purpose of this work is using numerical simulation methods to analyze the hydrodynamics and heat transfer in a channel with intensification in the form of a semi spherical depressions.

SOLUTION OF THE EXAMINED PROBLEM

Computer simulation was performed using the selected package of CAD/CFD programs SolidWorks/FlowSimulation.

In the CAD program SolidWorks, was constructed three-dimensional computing model of discretely roughed pipe with the following dimensional characteristics: Inner pipe diameter $d_b = 0.018$ m, external diameter $d_{out} = 0.022$ m, length $L = 1.6$ m.

As a discretely rough surface in the pipe were chosen spherical depressions (dimples) with sharp entrance and exit edges located in the in-line order with the pitch ratio between the dimple axes $S/h = 10$ and the dimple position angle $\varphi = 120$ °.

Following boundary conditions were set: at the pipe inlet – flow velocity range $V_{inlet} = 0.3$; 0,6 and 0,9 m/s with fluid temperature $t_{inlet} = 20$°C. Initial Reynolds numbers are $Re_{inlet} = 5000$, 10000, 16000. At pipe outlet: pressure $P_{outlet} = 101235$ Pa. The channel walls have physical properties of aluminum and are heated up to the temperature t_{wall}=105°C. Fluid flow is turbulent. For the mathematical simulation of the medium motion and heat transfer are used the non-stationary equations of Navier-Stokes: the energy equation (the first law of thermodynamics) and the equation of state [10-11].

For turbulent flows, the initial equations are averaged by the Reynolds method taking into consideration additional intensity caused by turbulent fluctuations of the parameters [11].

Incomplete system of equations, which was obtained based on above, is closed with the assistance of additional equations for the kinetic energy of turbulence - k and the dissipation of turbulence energy - ε in accordance with the well-known k - ε model of turbulence [11].

For the numerical solution of the problem, the initial system of nonstationary Navier-Stokes equations with additional equations describing the turbulent transfer is discretized in terms of space in the computational area and time.

As a result, the entire computational area is covered by a computational grid: the size and number of grid cells are determined by the user or automatically.

The finite volume method is used to discretize the differential equations and solve the resulting system of algebraic equations in the FlowSimulation program.

To gain the satisfactory accuracy of the solution results in this work were required about 1,000,000 to 1,200,000 liquid and solid elements depending on the type of the problem.

RESULTS AND DISCUSSION

Based on the results of numerical calculation was considered movement of a liquid in longitudinal section, on a site of discretely rough channel in the field of depressions.

The flow of liquid moves along a smooth channel with a constant velocity V. Reaching the inlet edge of the depressions, the flow moves without changing direction of its motion. And when the fluid flow reaches the outlet edge of the dimple, it turns back around dimple, forming the so-called vortex. Then it turns back to the outlet edge, while still retaining its full energy.

It is important to note, that at slowdown of returnable flow along a wall dimple the liquid heats up, before this current will get in area of the mixed flows.

The obtained result of numerical simulation does not contradict the experimental studies carried out in the publications [10].

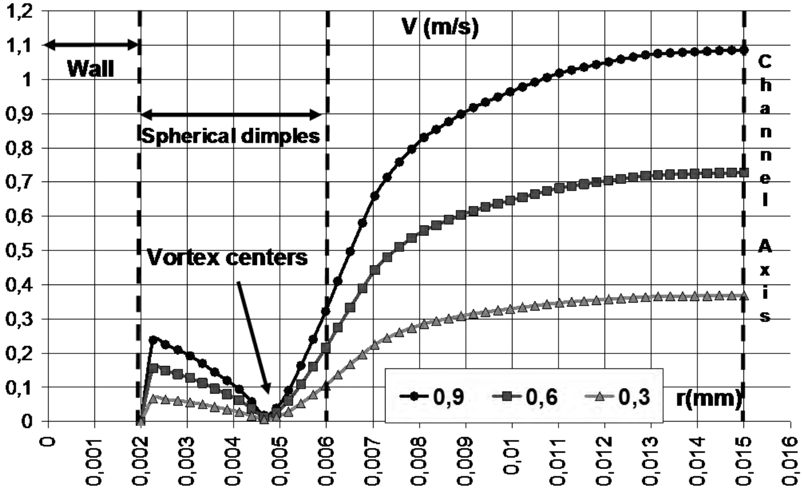

Fig.1. The dependency graph of the local velocity of fluid flow against the radius of the channel. Circle-V_{inlet} = 0,9 m/s, square - V_{inlet} = 0,6 m/s, triangle -V_{inlet} = 0,6 m/s.

A complex vortex structure exists in the dimple. As a result, velocity decreasing in the bottom of the dimple (near the wall). When current receding from the walls its velocity suddenly increases, and upon approaching the center of the vortex, the velocity decreases When moving further from the center of a vortex in dimple outlet direction, sudden increase fluid flow velocity is observed once again. However near to the channel axis, the flow velocity is not affected by significant changes.

The readings were taken at the set points, along the radius of the direct channel and the dimple. The breakdown of the local velocities of the flow at the points under consideration are shown in Fig.1.

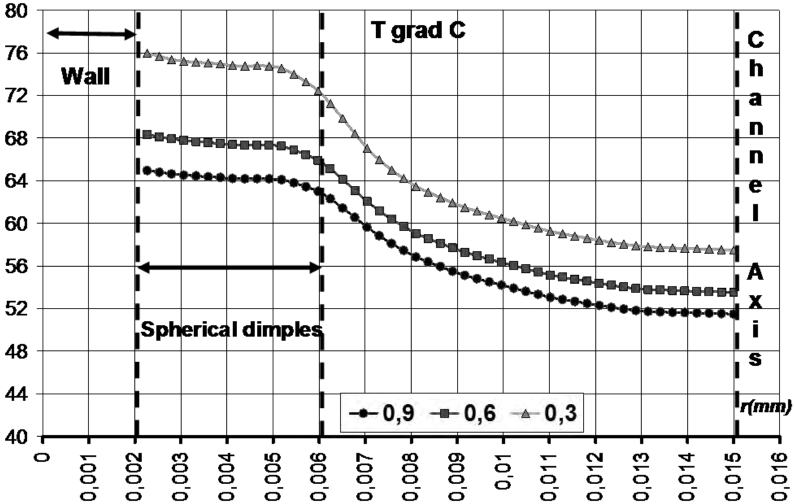

Fig. 2. the graph shows of the temperature dependency on the radius. Circle-V_{inlet} = 0,9 m/s, square - V_{inlet} = 0,6 m/s, triangle -$V_{inlet\,=}$ 0,6 m/s.

The obtained results of temperatures breakdown show, that in area dimples the liquid heats up more intensively at lower velocity of the currents (see fig.2).

At the outlet of the dimples thermal layers begin to mix with other currents and liquid temperature begins to decrease. Insignificant temperature changes are observed closer to the axis of the channel, which can be explained by the fact, that near the axis of the channel less intensive mixture of the liquid layers takes place. The graph of the temperature dependency on the radius is shown in Fig. 2.

The intensity of heat exchange which is described by dimensionless index of heat exchange Nu (Nusselt number), was defined by the formula (1):

$$Nu = 0{,}021 \cdot \mathrm{Re}_{water}^{0,8} \cdot \mathrm{Pr}_{water}^{0,43} \cdot \left(\frac{\mathrm{Pr}_{water}}{\mathrm{Pr}_{wall}}\right)^{0,25} \qquad (1)$$

where, Nu_1 – heat exchange coefficient, Pr (Prandtl number) – index of liquid and wall; $Pr = \mu C_p/\lambda$, were C_P – thermal capacitance at constant pressure, λ - thermal conductivity index, μ – dynamic viscosity index.

Reynolds number included in equation (1) is determined by the formula (2):

$$\mathrm{Re} = \frac{\rho \cdot V \cdot r}{\mu} \qquad (2)$$

where, Re - Reynolds number, r – radius (m), V – velocity of fluid flow, μ – dynamic viscosity index, $\rho_{1,2}$ – density of the flow.

All parameters included in the equation (1, 2) were obtained based on the results of numerical calculation in the set points.

The analysis of the obtained results indicated, that the Pr (Prandle criterion), which characterizes the physical properties of the liquid, included into Equation (1), does not have a significant impact on the value of the Nu (Nusselt number).

Nu (Nusselt number) is mostly impacted by the velocity parameter included in the dimensionless Reynolds number due to the vortex formations in the dimples.

With an increase in the flow velocity in a channel with a discrete roughness in comparison to a smooth one, the Nusselt number increases for more than 20%. The results of the calculation are shown in Fig. 3.

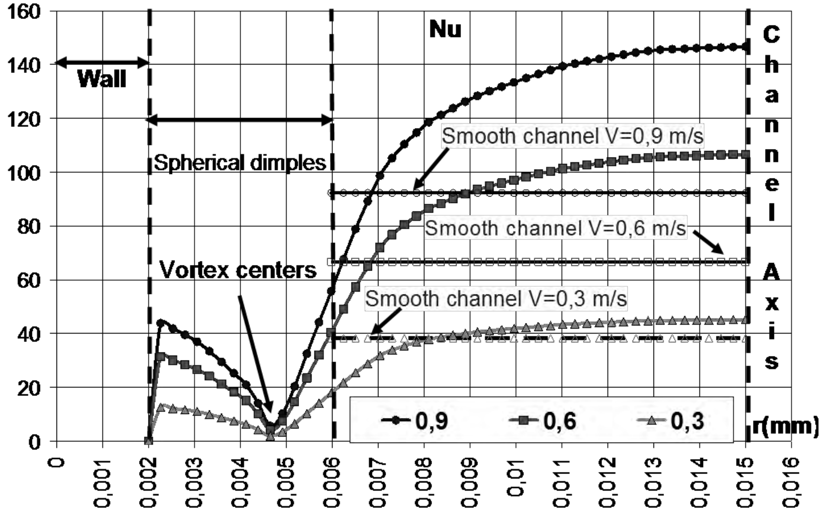

Fig.3. Graph of dependency of Nusselt number versus channel radius. Circle-V_{inlet} = 0,9 m/s, square - V_{inlet} = 0,6 m/s, triangle -V_{inlet} = 0,6 m/s. Straight lines are the average Nusselt number in a smooth channel for the considered flow velocity.

CONCLUSION

In the work conducted the Numerical simulation of hydrodynamics and heat transfer in a round channel with a discrete roughness made in the form of semi spherical dimples.

Based on the obtained calculated data is determined the Nusselt number, which characterizes the dimensionless index of heat exchange. It is shown that the heat exchange in the channels under consideration depends on the presence of the reverse flow currents zones formed in the dimples.

Assessment of results for the smooth and discrete rough channels showed, that with the flow velocity increase in the discrete rough channel the Nusselt number increasing by more than 20% in comparison to the smooth channel.

REFERENCES

[1] Mahmood G. I., Hill M. L., Nelson D. L., Ligrani P. M., Moon H.-K., Glezer B. Local heat transfer and flow structure on and above a dimpled surface in a channel. Transactions of the ASME. *Journal of Turbomachinery*, 2001,vol. 123, iss. 1, pp. 115 - 123. DOI: 10.1115/1.1333694.

[2] Mahmood G. I., Ligrani P. M. Heat transfer in a dimpled channel: ombined influences of aspect ratio, temperature ratio, Reynolds number, and flow Structure. *Intern. J. of Heat and Mass Transfer*, 2002, vol. 45, iss. 10, pp. 2011-2020.

[3] Kiselev N.A., Burtsev S.A., Strongin M.M. A procedure for determining the heat transfer coefficients of surfaces with regular relief. *Measurement Techniques*, 2015, vol. 58, no. 9, pp. 1016-1022.

[4] Кикнадзе Г. И., Олейников В. Г. Самоорганизация смерчеобразных вихревых структур в потоках газов и жидкостей и интенсификация тепло- и массообмена. –Новосибирск: Препринт 227-90 Институт теплофизики СО АН СССР, -1990. 45 с.

[5] Александров А.А., Горелов Г.М., Данильченко В.П., Резник В.Е. Теплоотдача и гидравлическое сопротивление при обтекании поверхности с развитой шероховатостью в виде сферических углублений // *Пром.теплотехника*. - 1989.-11,№6.-С.57-63.

[6] ChapmanD. R., A Theoretical Analysis of Heat Transfer in Regions of Separated Flow, NACA TN 3792, Oct. 1956.

[7] Хаген Р. Л., Данак А. М. Теплообмен в области отрыва турбулентного пограничного слоя при обтекании впадины// *Теплопередача*.- 1967.- №4.- С.62-69.

[8] Халатов А.А. Теплообмен и гидродинамика около поверхностных углублений (лунок) – Киев: НАН Украины, Институт технической теплофизики, 2005. -59 с.

[9] Sidenko N., Dzelzitis E. The method of numerical modeling of hydrodynamics and heat exchange in a channel with discrete roughness// *International Scientific Journal „Mathematical Modeling" year II*, 2018, 61-65 pp.

[10] Chang P. Separation of Flow //New York, Pergamon Press, 1971.

[11] Алямовский А.А. и др. SolidWorks 2007/2008. Компьютерное моделирование в инженерной практике.// БХВ– Петербург, Санкт – Петербург, 2008.

This work has been supported by the European Regional Development Fund within the Activity 1.1.1.2 "Post-doctoral Research Aid" of the Specific Aid Objective 1.1.1 "To increase the research and innovative capacity of scientific institutions of Latvia and the ability to attract external financing, investing in human resources and infrastructure" of the Operational Programme "Growth and Employment" (No Nr.1.1.1.2./VIAA/1/16/093).