TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
MECHANICS | MECHANIKA
2-M/2015

HASSAN MOMENI*, MARIUSZ DOMAGAŁA**

CFD SIMULATION OF TRANSPORT SOLID PARTICLES BY JET PUMPS

SYMULACJA CFD TRANSPORTU CZĄSTEK STAŁYCH W POMPACH STRUMIENICOWYCH

A b s t r a c t

This paper presents CFD simulation of solid particle transport by jet pump. Simulation was conducted using Lagrangian-Eulerian CFD approach. There is presented simulation of transport of mixture of water and solid particles with various density by jet pump with annular motive nozzle. The CFD simulations allowed to obtain trajectory for individual particles as well as theirs maximal velocity.

Keywords: *jet pump, CFD, particle tracking simulation*

S t r e s z c z e n i e

W artykule przedstawiono symulację transportu cząstek stałych w pompach strumienicowych przy wykorzystaniu narzędzi CFD. Symulacje przeprowadzono dla pompy z dyszą centralną przy rożnych gęstościach ciał stałych. Przeprowadzone symulacje pozwoliły uzyskać trajektorie oraz maksymalne prędkości cząstek stałych podczas pracy pompy.

Słowa kluczowe: *pompa strumienicowa, analiza CFD, transport cząstek stałych*

* Ph.D. Hassan Momeni, Institute of Machine and Maritime Technology, Bergen University College.
** Ph.D. Mariusz Domagała, Institute of Applied Informatics, Faculty of Mechanical Engineering, Cracow University of Technology.

1. Introduction

Jet pumps are widely used in many industry branches like chemical, petrochemical, energy and even in food industry. Among many advantageous of jet pumps the most important are: no moving parts, simplicity, reliability and relatively low cost. Such features cause that jet pumps are used for transport of extremely hazardous liquids, slurries or even mixture of liquids and solid particles [1]. There is available a lot of design of jet pumps, but the main group we can distinguish is pump with circumferential or central nozzle. For the transport of solid objects the more convenient is to use jet pump from the first group. Jet pumps can also be distinguish depending on the driving and driven fluid. The jet pump in which both streams are liquids is knows as Liquid-Jet-Liquid pump. Such pumps can be used also for transport liquids which contains solid particles. Simulation of jet pump is not an easy task, even in case of LJL pumps when both streams are homogenous and in the same state conditions. It is even more difficult when we have to simulate transport of liquids which contain solid particles. Such simulations are possible with the use of Computational Fluid Dynamics method and Lagrangrian-Eulerian approach [2].

2. Mathematical modeling of solid particles in CFD

Flow of fluid through jet pump is governed by well known Navier-Stokes equations, which in CFD methods are solved numerically. In case of mixture with solid particles we have to add additional particle transport model. As the CFD use Eulerian approach for solving fluid flow one of the way to describe particle tracking is the Lagrangian approach. This involves the integration of particle paths through the discretized domain. Individual particle is injected to fluid domain and an additional source term is added to fluid mass, and momentum equations. The equation of motion for individual particle can be written as [3]:

$$m_P \frac{d\mathbf{U}_P}{dt} = \mathbf{F_D} + \mathbf{F_B} + \mathbf{F_R} + \mathbf{F_{VM}} + \mathbf{F_P} \tag{1}$$

where:

- $\mathbf{F_D}$ – drag force at the particle,
- $\mathbf{F_B}$ – buoyancy force,
- $\mathbf{F_R}$ – centripetal and Corliolis force,
- $\mathbf{F_{VM}}$ – virtual mass force, this force is caused by accelerating some of surrounding fluid by the particle,
- $\mathbf{F_P}$ – is the pressure force.

The drag force $\mathbf{F_D}$ on a particle is proportional to the slip velocity between the particle and fluid velocity:

$$\mathbf{F_D} = \frac{1}{2} C_D \rho_F A_F \left|\mathbf{U_F} - \mathbf{U_P}\right| \left(\mathbf{U_F} - \mathbf{U_P}\right) \tag{2}$$

The buoyancy force $\mathbf{F_B}$ is:

$$\mathbf{F_B} = \frac{\pi}{6} d_P^3 \left(\rho_P - \rho_F\right)\mathbf{g} \tag{3}$$

The centripetal and Corliolis force force $\mathbf{F_R}$ is:

$$\mathbf{F_R} = m_P(-\mathbf{\Omega} \times \mathbf{U_P} - \mathbf{\Omega} \times \mathbf{\Omega} \times \mathbf{r_P}) \tag{4}$$

The virtual mass force $\mathbf{F_{VM}}$ can be expressed as:

$$\mathbf{F_{VM}} = \frac{C_{VM}}{2} m_F \left(\frac{d\mathbf{U_F}}{dt} - \frac{d\mathbf{U_P}}{dt}\right) \tag{5}$$

The pressure force $\mathbf{F_P}$ is:

$$\mathbf{F_P} = m_P \frac{\rho_F}{\rho_P}\left(\mathbf{U_F}\nabla\mathbf{U_F} - \mathbf{R_F}\right) \tag{6}$$

$$\frac{d\mathbf{U}_P}{dt} = \frac{1}{m_P + \frac{C_{VM}}{2} m_F}\left(\mathbf{F_D} + \mathbf{F_B} + \mathbf{F_R} + \mathbf{F'_{VM}} + \mathbf{F_P}\right) + \frac{1}{m_P}\mathbf{F_R} \tag{7}$$

For steady state conditions:

$$\mathbf{F'_{VM}} = \frac{C_{VM}}{2} m_F \left(\mathbf{U_F}\nabla\mathbf{U_F}\mathbf{F_R} - 2\mathbf{\Omega} \times \mathbf{U_F} - \mathbf{\Omega} \times \mathbf{\Omega} \times \mathbf{r_P}\right) \tag{8}$$

where:

$m_F = \frac{\pi}{6} d_P^3 \rho_F$ – mass of fluid surrounding the particle, d_p is the particle diameter,

$m_P = \frac{\pi}{6} d_P^3 \rho_P$ – mass of the particle,

$R_F = -2\Omega \times \mathbf{U_F} - \mathbf{\Omega} \times \mathbf{\Omega} \times \mathbf{r_P}$,

C_{VM} – coefficient,

$\mathbf{U_F}$, $\mathbf{U_P}$ – fluid and particle velocity, respectively,

$\mathbf{g}$ – gravity vector,

$\mathbf{\Omega}$ – computational frame angular velocity.

2. Simulation model

Numerical simulation was conducted in ANSYS CFX for model presented on Fig. 1. For simulation purposes only a half model was used, on the symmetry plane an appropriate boundary conditions were applied.

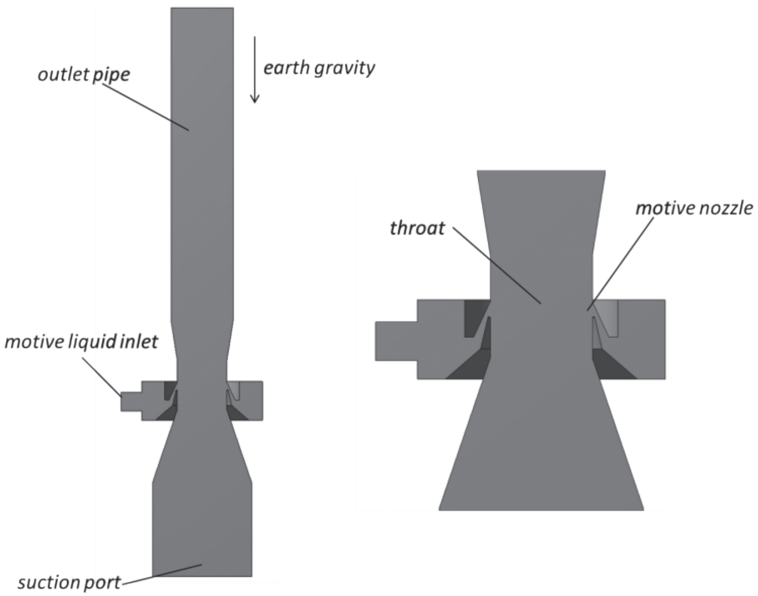

Fig. 1. Model of LJL pump used in CFD simulation

The diameter of suction port is 160 mm, where the solid particles are injected. Solid particles have 5 mm diameters and have possibility of collision of each other and with pump walls. During simulation constant fluid velocity at motive inlet was defined, it was 15 m/s, while density of solid particles were changed from density equal to water (for very light rock particles) to density of 3500 kg/m^3, what corresponds to heavy rocks. Simulations were conducted for steady state conditions, for turbulent flow at thermal equilibrium state. No erosion or cavitation was included in simulation model.

3. CFD results

Results of the CFD simulations are presented on Figs 2–4, which shows particle trajectories for identical flow conditions and diameter of particles but with various density of solid particles. Solid particles varies from ratio $r = 1$ to $r = 3.5$, where $r = \frac{\rho_P}{\rho_F}$, ρ_F is water density, while ρ_P is particle density.

Results presented on Fig. 1 shows that the particle velocity is not uniform and some swirls appears after the particles passing the throat what might be caused by non uniform fluid velocity at this region. Furthermore, the maximal velocity of solid particles is almost 1/3 than fluid velocity. Maximal velocity of particles is reached after they went through the throat.

Next figures shows trajectory for particles with higher density. When comparing results presented on Fig. 2, Fig. 3 and Fig. 4 we may noticed that as the particles density rises the

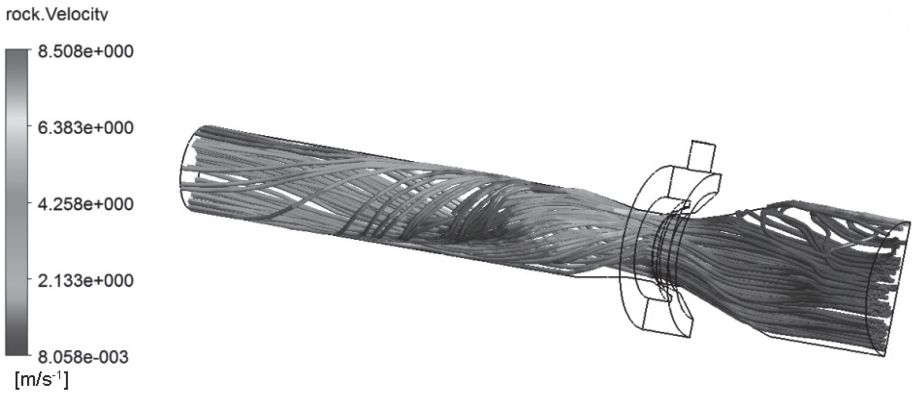

Fig. 2. Solid particles trajectory for density ratio $r = 1$

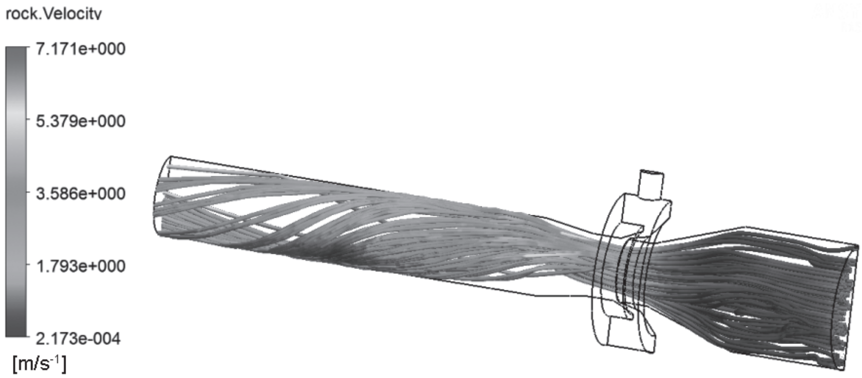

Fig. 3. Solid particles trajectory for density ratio $r = 2$

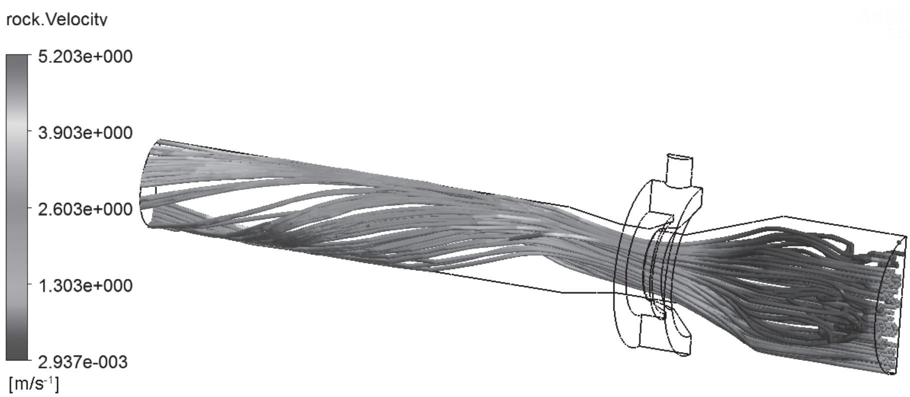

Fig. 4. Solid particles trajectory for density ratio $r = 3.5$

amount of particles which is taken away by fluid decrease. It can be also found that the trajectory of particles changes as the density of particle increase. And also maximal particle velocity drops from 8.5 m/s to 5.2 m/s.

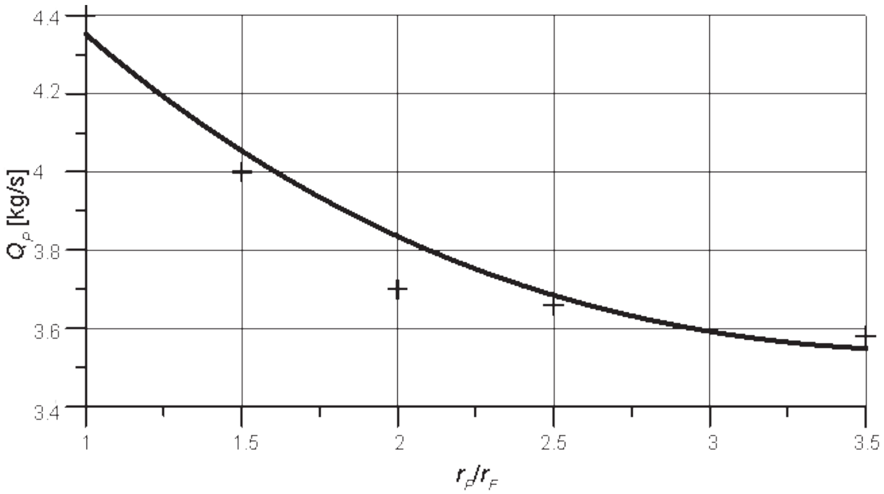

Fig. 5. Mass flow rate of solid particles depending on particle diameter

Figure 5 shows the mass flow rate at the pump outlet for various particles density and reach values of 4.4 kg/s for particle density equal to water (r = 1) to value 3.58 kg/s for density ratio r = 3.5.

4. Conclusions

This article presents CFD simulation of transport solid particles by jet pump using Lagrangian-Eulerian approach. Simulations were conducted for various particle density what allowed to investigate particle motion during fluid flow. CFD analysis allowed to investigate particles trajectory depending on theirs weight as well as obtain quantitative information like mass flow rate of solid particles on pump outlet. CFD simulation allowed also to obtain information about maximal particles velocity, which was found to be more than three times less that driven fluid velocity. Furthermore, it has been found that the particle velocity is not uniform within the jet pump cross section.

CFD methods with applied Lagrangian-Eulerian approach seems to be very efficient tool in simulation of transport mixtures of liquids and solid particles.

References

[1] Karassik I.J., Messina J.P., Cooper P., HealdCh.C., *Pump Handbook*, Third Edition, McGraw Hill, 2001.

[2] Xue-Guan Song, Joon-Hong Park, Seung-Gyu Kim, Young-Chul Park, *Performance comparison and erosion prediction of jet pumps by using a numerical method*, Mathematical and Computer Modelling, Volume 57, Issues 1–2, January 2013, 245-253.

[3] Ansys CFX Theory Guide.

[5] Domagala M., *Simulation of cavitation in jet pumps*, Technical Transactions. Mechanics, Kraków 2013, 51-58.

[6] Momeni H., *CFD modelling of two stages liquid jet liquid pump*, Rozprawa doktorska, Politechnika Krakowska, Kraków 2011.