International Journal of Heat and Mass Transfer 55 (2012) 7645–7650

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical investigation of the drag force reduction induced by the two-phase flow generating on the solid body surface

Jonas Gylys *, Linas Paukštaitis, Raminta Skvorčinskienė

Kaunas University of Technology, K. Donelaičio 20, 44239 Kaunas, Lithuania

ARTICLE INFO

Article history:
Received 20 June 2012
Accepted 25 July 2012
Available online 22 August 2012

Keywords:
Drag force reduction
Hot body
Two-phase flow
ANSYS CFX code

ABSTRACT

An article presents an analysis of the drag force reduction during the hot spherical and cylindrical body movement in the water in comparison with the movement of the cold bodies. Influence of two-phase water–vapor flow on the drag was investigated experimentally and numerically using the ANSYS CFX code. The preliminary results of the investigation showed a sufficient reduction of the drag force for the hot body in comparison to the cold body due to the vapor presence.

1. Introduction

During the movement of the solid body in the fluid (liquid or gas) a drag force occurs, which affects on the stability of the movement and on the energy consumption necessary for body's motion.

The drag force depends on the fluid characteristics, on the solid body cross-sectional area (shape), etc. Particularly, the drag force is influenced by fluid density – the lower density, the lower drag force. In addition, resistance force is as bigger, as much the solid body velocity is higher. As a result, it is necessary to reduce the drag force, which effects the solid body movement [1].

The resistance to the solid body movement in a liquid can be reduced not only by changing the solid shape, but also by reducing density of the environment (liquid), in which the solid body moves [2]. In addition, there is no need to reduce the density of total fluid volume, it is sufficient to cover up the moving body with less density layer of liquid or gas. Thus, the frontal and friction resistance will be reduced accordingly.

The lower density of the environment (gas) layer around the moving body can be created in many different ways. For example, it can be made by using special holes, fitted in the front part of the moving solid body, let/feed gas, which can cover the moving body by gas (air, combustions products) envelope.

Another way is to use a super cavitation effect [3]. A super cavitation can be created on the solid bodies, which has a specific geometry. The front surface of such bodies is flat with sharp prominence sideways. Then the solid body moves in a liquid at a high velocity, a fluid is compressed on a flat front of the body. Behind that surface the pressure falls down suddenly. Due to the big difference of the pressures a gas pillow occurs at the solid body surface. Super cavitation effect is possible only at high velocities of the solid body (approximately 180 km/h and more). At the low velocities gas pillow generates slowly, thus the drag force changes a little only. In order to increase and stabilize this process, the combustion products are injected to the liquid.

Another method, which helps to form a gas (steam) envelope, is based on the second order boiling crisis fact. On the surface of a solid body, heated to the high temperatures, usually forms an integral gas (steam) film, which surrounds the moving solid body and reduces the drag force also [4].

The conditions of the fluid and the gas layer change then the fluid flows around the heated surface of the solid body. If the temperature of fluid grows, the density of it reduces and the phase transformation may occur.

The aim of this investigation is to model the resistance (drag force) reduction for the case, when one-phase (water) flow turns to the two-phase flow on the heated surface of the moving solid body.

There were used two solid bodies with a different shape: sphere and cylinder. An article presents the preliminary results of that investigation.

2. Governing equations in ANSYS CFX code

2.1. Turbulence

It is important to choose the appropriate turbulence model for the numerical investigation. Several different turbulence models

* Corresponding author.
E-mail addresses: jonas.gylys@ktu.lt (J. Gylys), linas.paukstaitis@ktu.lt (L. Paukštaitis), raminta.skvorcinskiene@gmail.com (R. Skvorčinskienė).

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.07.078

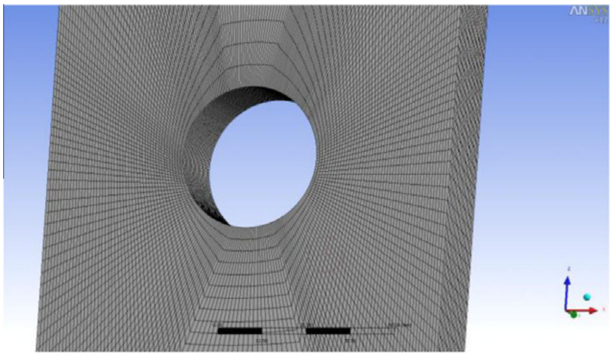

Fig. 1. Calculating grid for the test of turbulence models.

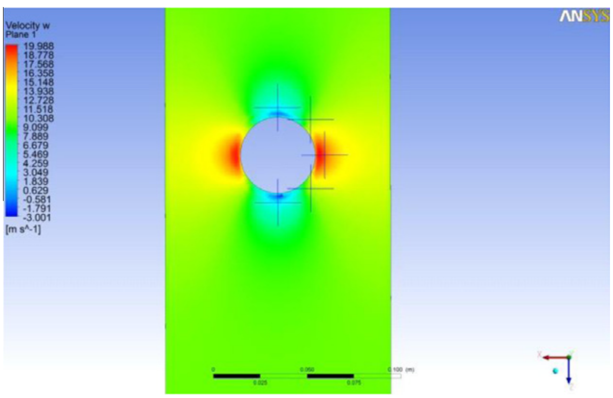

Fig. 2. Distribution of the turbulence model test results.

can be applied. A few of them are more accurate for the high flow velocity case. Other models are adapted for the calculation of the boundary layer. It is important to select such a turbulence model which could more precisely reflect the experimental data.

Turbulence is a difficult and complex no stationary process, which runs in all directions at a flow. Turbulence process begins at the time then the inertia forces in fluid become significantly higher in comparison to the viscosity forces.

Usually the turbulence models of the program packages (codes), which are used for modeling of the fluid flows, are based on the statistical data.

The ANSYS CFX code provides a possibility to use five different turbulence models: k–ε, k–ω, SST, BSL and SSG. Each of those models has different hypotheses and equations that describe these models. For instance, k–ε model assumes, that the turbulent viscosity is associated with the kinetic energy and diffusion of turbulence, while the k–ω – turbulent viscosity is related to the kinetic energy and frequency of turbulence. Few of mentioned turbulence models are in addition to the models k–ε and k–ω (BSL model combines the advantages of k–ε and Wilcox) [5].

In order to test the turbulent models, the numerical model (a cylinder in the channel) was developed. Dimensions of the cylinder: diameter is 0.04 m, length is 0.023 m. Dimensions of the channel: width is 0.2 m, length is 0.023 m, height is 0.5 m. The air flows through the channel, flow velocity is 10 m/s. Fig. 1 shows the calculating grid made by the ANSYS CFX code.

Five points on the surface of cylinder are marked in Fig. 2. Points are located clockwise. Test results are presented at Table 1.

As can be seen, turbulence modeling results are fairly good and confirm to the experimental data. k–ε model was selected for the further calculations, as more suitable for modeling turbulent flow at the small flow velocities.

Table 1
The test results of ANSYS CFX code turbulence model. Experimental results are taken from the [6].

Point	1	2	3	4	5
Experiment	−2.64	3.17	14.8	10.9	2.31
k–ε	0.84	4.69	17.72	11.35	13.29
(sk-eks)/eks	−1.32	0.32	0.16	−0.93	−0.68
SST	0.526	5.23	16.02	8.88	11.43
(sk-eks)/eks	−1.2	0.65	0.08	−0.19	3.95
BSL	0.693	5.17	16.01	8.73	11.42
(sk-eks)/eks	−1.26	0.63	0.08	−0.2	3.94
SSG	−0.689	4.809	15.56	8.53	9.745
(sk-eks)/eks	−0.739	0.517	0.051	−0.22	3.22

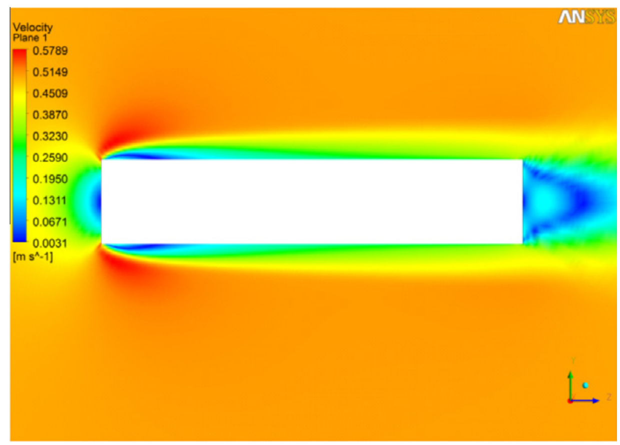

Fig. 3. Water flow around the cylindrical body.

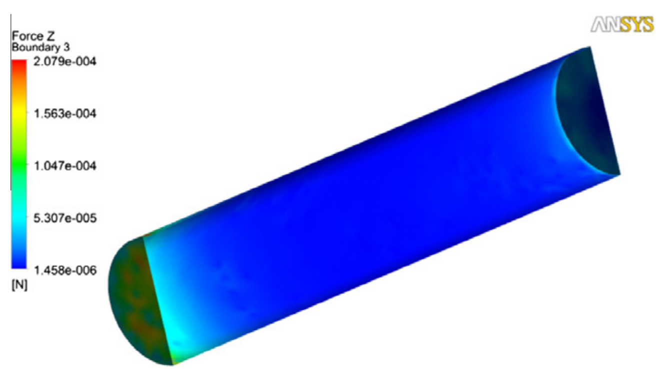

Fig. 4. Drag force changes for the cylindrical body, moving in the water flow.

The k–ε model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation [5]:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}, \tag{1}$$

where μ_t – the eddy viscosity or turbulent viscosity, which must be modeled using k–ε model, C_μ is a constant equal to 0.09 and k – the turbulence kinetic energy; ε is the turbulence eddy dissipation.

The values of k and ε come directly from the differential transport equations [5]:

$$\frac{\partial(\rho k)}{\delta t} + \nabla(\rho U k) = \nabla\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right) + P_k + P_{kb} - \rho\varepsilon, \tag{2}$$

and

$$\frac{\partial(\rho \varepsilon)}{\delta t} + \nabla(\rho U \varepsilon) = \nabla\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right) + \frac{\varepsilon}{k}(C_{\varepsilon 1}(P_k + P_{\varepsilon b}) - C_{\varepsilon 2}\rho\varepsilon), \tag{3}$$