Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 85 (2016) 260 – 265

Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO - YRC 2015, 18-20 November 2015, Bucharest, Romania

Numerical study for open-channel flow over rows of hemispheres

Elena Iatan[a]*, Marius Iliescu[a], Florin Bode[b*], Ilinca Nastase [a], Radu Mircea Damian[a], Mihnea Sandu[a]

[a]*Technical University of Civil Engineering in Bucharest, Building Services Department, 66 Avenue Pache Protopopescu; 020396, Bucharest, Romania*
[b]*Technical University of Cluj-Napoca, Cluj-Napoca, Romania*

Abstract

Important parameters in the study of design criteria for sewer pipes are related to shear velocity. The aim of this paper is to obtain a numerical model which does not require significant computational resources and that is feasible for the calculation of shear velocity. We created a numerical model in a free surface flow over artificial roughness elements represented by hemispheres. The numerical model is designed to be identical with an experimental model previously studied. We have investigate velocity and vorticity fields and the distribution of turbulent stresses using a SST k − ω model.

Peer-review under responsibility of the organizing committee EENVIRO 2015
Keywords: Artificial roughness; hemispheres; *κ-ω* SST model; particle image velocimetry

1. Introduction

It is of practical interest to understand and evaluate the effects that the presence of additional roughness will generate in the water flowing in a sewer pipe. Various experimental and numerical studies were performed to investigate flow characteristics of artificially roughened channels. In this study the additional roughness is represented by periodic rows of hemispheres with the length-to depth ratio equal to 4. The water flowing in this channel is numerically investigated using k-omega shear stress transport (k-ω SST) turbulence model, on a three-dimensional (3-D) domain.

* Corresponding author.
E-mail address: eserban@instal.utcb.ro; florin.bode@termo.utcluj.ro

Peer-review under responsibility of the organizing committee EENVIRO 2015
doi:10.1016/j.egypro.2015.12.242

An important method used for the study of flows over different roughness elements is the Computational Fluid Dynamics method (CFD). The flow filed is completely described by Navier–Stokes equations that are non-linear partial differential equations. Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are used for numerically solving Navier–Stokes equations, providing the most accurate solutions. Comparative results between DNS and LES are found in literature and in some circumstances LES is performing almost “like DNS”, see [1]. D. Chatzikyriakou et al. (2015) performed a DNS vs LES comparative study and determined that in LES simulations the values of turbulent kinetic energy is equal to at least 94% of the values of the kinetic energy computed in the DNS simulation [2]. The disadvantage of these two techniques (DNS and LES) is that they are requiring significant computational resources.

Most of the numerical studies of the fluid flow over roughness elements use the Reynolds-Averaged Navier-Stokes (RANS) method with different turbulence models. RANS models, although they are introducing numerical and physical approximations, can perform reasonably accurate with less computational resources. Numerous studies for different wall-mounted obstacles have been performed by using turbulence models like standard k–ε model, modified k–ε model, Reynolds Stress Model, DNS and LES. Previous studies [3] have shown that k–ε RANS models can have problems in simulating the details of the flow over arrays of cubes. In particular they can overestimate turbulence energy upstream of stagnation points and do not capture accurately separation and reattachment processes. On the other hand, more accurate LES simulations can reproduce such details of the flow, having the disadvantage of a more expensive time calculation (e.g. Cheng et al. 2003).

Xie and Castro (2006) performed a numerical study including a comparison between LES and RANS emphasizing that the latter is inadequate, especially within the canopy region. The proposed RANS models in their study (SKE, MKE, RSM) underestimate the stream-wise mean velocity within the canopy and both the SKE and MKE models fail to predict a reverse flow on the lateral sides of the cube. All the RANS models underestimate the TKE in the canopy, when compared with the LES results. As a general conclusion SKE performed the worst.

Peng et al. (2011) studied the thermal/hydraulic performance of different ribs, also in terms of Nusselt number and friction factor, with the shear-stress transport model, k-ω SST. The numerical results for friction factor are observed to be much larger than the experimental data. A good agreement between numerical and experimental results is observed for the variation of the Nusselt number.

Tang and Zhu (2013) investigated the turbulent flow and heat transfer in a rectangular channel with inclined broken ribs. Numerical results are evaluated by comparison with experimental data for Nusselt number and friction factor and they concluded that the k-ω SST model is more suitable than RNG k − ε model for numerical simulation of flow and heat transfer in ribbed channels. Differences between experimental and numerical results for Nusselt number and friction factor are less than 10% and 7% respectively.

The main objective of the present study is to investigate and determine the range within which the numerical k-ω SST model responds correctly, compared to the experimental data, for the geometry presented in Fig. 1. The authors investigated velocity fields and the Reynolds stress distribution. Shear velocity is a critical element in the studies concerning flows over rough surfaces and can be calculated from the velocity distributions or Reynolds stresses. The procedures are not presented in this paper, for references see [6], [7], [8].

2. Experimental measurements

The experiments were performed in a closed loop water system of pipes. A short geometrical description is presented in Fig. 1 and considers a three-dimensional free surface flow of water over wall-mounted hemispheres at the bottom wall of the channel.

The channel is circular with a diameter of 14 cm. The water depth H is equal to 68 mm. The diameter of the hemispheres Φ is 4,5 mm and the height of the hemisphere h is 2,25mm. The water depth is corresponding to 27 times the hemisphere height (H/h=68/2,25=30). The section in which the measurements were made is positioned at 3,5m from the water inlet and it includes three consecutive hemispheres (in the flow direction) along the row R3 (see Fig. 1). The rows are made of calibrated hemispheres attached to the bottom model surface.

The flow rates in the supply pressure pipe were measured by a volumetric flow meter with accuracy of 3%. The measured water temperature was 22°C + - 10C, for which the corresponding kinematic viscosity was taken 0,93 (m2/s) x 10-6. The bulk Reynolds number of the flowing water, based on the hydraulic radius, was 16,000.