The XXXVI International School of Hydraulics, 23-26 May 2017, Jachranka, Poland

A CFD based comparison of mixing due to regular and random cylinder arrays

M. GOLZAR[1], F. SONNENWALD[1], I. GUYMER[2] and V. STOVIN[1]

[1] Department of Civil and Structural Engineering, University of Sheffield,
Western Bank, Sheffield S10 2TN, UK
e-mail: mgolzar1@sheffield.ac.uk

[2] School of Engineering, University of Warwick
Library Road, Coventry CV4 7AL, UK

ABSTRACT

Numerous studies have focused on flow and mixing within cylinder arrays because of their similarity to vegetated flows. Randomly distributed cylinders are considered to be a closer representation of the natural distribution of vegetation stems compared with regularly distributed arrays. In this study the flow fields associated with two arrays of regularly and randomly distributed cylinders are modelled in two dimensions using ANSYS Fluent 16.1. The RSM turbulence model is used to model the turbulence closure, and all the variables are discretized using the second order upwind method. The resulting flow fields are used to run the solute transport model to characterise mixing within each geometry. For the same stem diameter and solid volume fraction, greater dispersion is evident in the random cylinder array compared with the regular array. Dispersion coefficient values are compared with those reported in the literature and a good agreement is shown. Turbulence length scales estimated from the velocity profiles and optimized dispersion coefficients are close to the cylinder diameter which is in agreement with theories in the literature.