Effects of computational meshes on hydrodynamics of an open channel junction flow using CFD technique

Adrien Momplot[1], Hossein Bonakdari[1,2*], Emmanuel Mignot[3], Gislain Lipeme Kouyi[1], Nicolas Rivière[3], Jean-Luc Bertrand-Krajewski[1]

[1]Université de Lyon, INSA Lyon, LGCIE - Laboratory of Civil & Environmental Engineering, F-69621 Villeurbanne cedex, France
[2]Department of Civil Engineering, Razi University, Kermanshah, Iran
[3]LMFA, CNRS-Université de Lyon, INSA de Lyon, Bat. Joseph Jacquard, 20 avenue A. Einstein, F-69621 Villeurbanne cedex, France
*Corresponding author, e-mail: bonakdari@yahoo.com

ABSTRACT

This paper deals with numerical calculation of flow structures in open-channel junction flows which are typical singularities encountered in urban drainage systems. The objective is to evaluate the impact of the mesh shape, mesh refinement, and free-surface modelling approach on the simulation. The ability of CFD strategy (appropriate numerical options, particularly: computational meshes, discretization schemes, turbulence models linked to wall treatment functions and air-water interface capturing approach) is determined by comparing simulated results against experimental one obtained on laboratory scaled open-channel junction where PIV measurement technic was set-up. Comparisons emphasis on simulated and measured horizontal velocity field, mixing interface between both inflows and recirculation zone at two elevations, near the free-surface and close to the bottom of the channel.

The mesh was refined until no significant changes are observed on the main flow structures. Nevertheless, whatever the mesh refinement, CFD modelling has some trouble to simulate accurately the recirculation zone extension near the bottom. The mesh shape has no influence on the velocity field even if hexahedral meshes give better representation of the mixing interface in the junction than tetrahedral meshes. Nonetheless, both mesh types do not enable to represent properly the recirculation in the near-bottom region. Finally, the use of the VOF model leads to a similar velocity field, but seems to give less ability to represent the mixing interface in the junction and the extension of recirculation zone.

KEYWORDS

Computational Fluid Dynamics, Hydrodynamics, Junction, Mesh size