Geophysical Research Abstracts
Vol. 16, EGU2014-11671, 2014
EGU General Assembly 2014

An evaluation of Computational Fluid dynamics model for flood risk analysis

Silvia Di Francesco (1), Chiara Biscarini (2), and Valeria Montesarchio (3)

(1) Niccolò Cusano, Rome, Italy (silvia.difrancesco@unicusano.it), (2) University of Foreigners of Perugia, Perugia, Italy (chiara.biscarini@unistrapg.it), (3) Niccolò Cusano, Rome, Italy (valeria.montesarchio@unicusano.it)

This work presents an analysis of the hydrological-hydraulic engineering requisites for Risk evaluation and efficient flood damage reduction plans.
Most of the research efforts have been dedicated to the scientific and technical aspects of risk assessment, providing estimates of possible alternatives and of the risk associated. In the decision making process for mitigation plan, the contribute of scientist is crucial, due to the fact that Risk-Damage analysis is based on evaluation of flow field ,of Hydraulic Risk and on economical and societal considerations.
The present paper will focus on the first part of process, the mathematical modelling of flood events which is the base for all further considerations.
The evaluation of potential catastrophic damage consequent to a flood event and in particular to dam failure requires modelling of the flood with sufficient detail so to capture the spatial and temporal evolutions of the event, as well of the velocity field. Thus, the selection of an appropriate mathematical model to correctly simulate flood routing is an essential step.
In this work we present the application of two 3D Computational fluid dynamics models to a synthetic and real case study in order to evaluate the correct evolution of flow field and the associated flood Risk .
The first model is based on a opensource CFD platform called openFoam. Water flow is schematized with a classical continuum approach based on Navier-Stokes equation coupled with Volume of fluid (VOF) method to take in account the multiphase character of river bottom-water- air systems.
The second model instead is based on the Lattice Boltzmann method, an innovative numerical fluid dynamics scheme based on Boltzmann's kinetic equation that represents the flow dynamics at the macroscopic level by incorporating a microscopic kinetic approach. Fluid is seen as composed by particles that can move and collide among them.
Simulation results from both models are promising and congruent to experimental results available in literature, thought the LBM model requires less computational effort respect to the NS one.