

CFD Modeling of Pollutant Dispersion in a Free Surface Flow

Authors : Sonia Ben Hamza, Sabra Habli, Nejla Mahjoub Said, Hervé Bournot, Georges Le Palec

Abstract : In this work, we determine the turbulent dynamic structure of pollutant dispersion in two-phase free surface flow. The numerical simulation was performed using ANSYS Fluent. The flow study is three-dimensional, unsteady and isothermal. The study area has been endowed with a rectangular obstacle to analyze its influence on the hydrodynamic variables and progression of the pollutant. The numerical results show that the hydrodynamic model provides prediction of the dispersion of a pollutant in an open channel flow and reproduces the recirculation and trapping the pollutant downstream near the obstacle.

Keywords : CFD, free surface, pollutant dispersion, turbulent flows

Conference Title : ICEWE 2015 : International Conference on Energy, Water and Environment

Conference Location : Venice, Italy

Conference Dates : November 09-10, 2015