CFD Study of Free Surface Flows Resulting from a Dam-Breaking

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

Abstract : Free surface flows caused by dam breaks in channels or rivers is an attention-getting subject to the engineering practice, however, the studies are few to be reported. In this paper, a numerical investigation of unsteady free surface flows resulting from a dam-breaking in a rectangular channel is studied. Numerical computations were carried out using ANSYS Fluent which is based on the finite volume approach. The air/water interface was modeled with the volume of fluid method (VOF). Verification for a typical dam-break problem is analyzed by comparing the present results with others and very good agreement is obtained. The present approach is then used to predict the characteristics of free surface flow due to the dam breaking in channel. The characteristics of complex unsteady free surface flow in these examples are clearly explained. The numerical results show that the flow became more disturbed after impacting the vertical wall, then a recirculation zone, as well as turbulence phenomena, were created. At this instant, a cavity of air was included on the flow. The results agree well with the experimental data found in the literature.

Keywords : CFD, dam-break, free surface, turbulent flows, VOF

Conference Title : ICFMTE 2016 : International Conference on Fluid Mechanics and Thermal Engineering

Conference Location : Istanbul, Türkiye

Conference Dates : April 19-20, 2016