Numerical simulation of flow in open channel with a 90^o bend

Baghalian S.[1, 2], Bonakdari H.[2, 3], Fazli M.[4]

1- M.Sc. student, Department of Civil Engineering, University of Razi, Kermanshah, Iran,
2- Water and Wastewater Research Center, University of Razi, Kermanshah, Iran,
3- Assistant professor, Department of Civil Engineering, University of Razi, Kermanshah, Iran,
4- Assistant professor, Department of Civil Engineering, University of Bu-Ali Sina, Hamedan, Iran,
S_baghalian@yahoo.com

Abstract

A new methodology for resolve complex problems in fluid mechanics has been known as computational fluid dynamics (CFD). CFD investigates parameters such as velocity field, pressure, shear stresses, effect of turbulence and etc in flow field. In this paper, a three-dimensional CFD model was used to investigate of flow patterns, velocity profiles and also accuracy of numerical models in 1phase and 2 phases was investigated. For numerical simulation ANSYS-CFX software was used. The *k-ε* turbulence model was used to solve turbulence equations. The results showed that secondary flow and centrifugal force influenced flow pattern and have good agreement with experimental.

Keywords: numerical simulation, Computational Fluid Dynamic, bend, velocity, CFX

1. INTRODUCTION

Studying of flow pattern in curved channels is one subject that researchers and engineers of hydraulic science attend it. One of more important characteristics in channels with bend is presence of secondary flow caused by centrifugal force. Presence of spiral flow in curved channels cause complex flow pattern in meandering rivers [1]. This type of flow influence sedimentation and erosion and make irregular topography in bend [2, 3, 4, 5]. Mosonyi and Gotz [6] were first persons that investigated effects of spiral current on flow field and manner of its variations in curved channels. During the twentieth century, with advent of the high-speed digital computer, CFD has developed. CFD use numerical techniques for solving governing equations of flow for given geometry and boundary conditions. This provides the possibility of more accuracy and simpler prediction of flow field. Only special problems of flow could be solved analytically and without computer [7]. Several studying is done about secondary flow and effect of it in flow field. Bathurst et al. [8] investigated secondary flow and shear stress in meanders. They showed secondary current have low intensity in high and low discharges and have high intensity in mean discharges. Lien et al. [9] using a 2D numerical model averaged in depth, studied effect of secondary current in calculation of dispersion stresses and flow pattern. One of important results obtained from their studies was role of dispersion stresses in prediction of flow filed in strong or weak bends. Also De Marchis and Napoli [10] using a 3D numerical model investigated flow field in curved open channels. This model solved 3D Reynolds-averaged Navier-Stocks equations and *k–ε* model had been used for solving turbulence equations. Won Seo and Jai Jung [11] investigated velocity distribution of secondary currents in channels with bend. They compared velocity profiles obtained from results of theory equations to experimental results. Then they did exact analysis using hydraulic parameters on theory equations. Demuren and Rodi [12] predicted the flow and transport of a neutral tracer in curved channels by using a three-dimensional model and *k–ε* model. They presented a predictive numerical method which solves the full, 3D, two-fluid model equations in dispersed two phase flow using control-volume discretization. Olsen [13] computed the bed variations in a 90°channel bend with special respect to steep transversal slopes. In another study, Nouh and Townsend [14] investigated distribution of shear stress in stable curved channels. They showed that secondary currents caused local asymmetry in the flow field and these secondary currents had been made by stream curvature in channel bends. Also Wu et al. [15] by using 3D numerical model, studied flow field and sediment transport in channel bends. In another study Huang et al. [16] recommended a method to compute 3D sediment transport effect in channel with bends. Dietrich and Smith [17] investigated topography of river bed using mathematical model. They showed that spiral current cause transfer of maximum shear stress toward outer bend, so maximum scour will occur in that place. Also Duan et al. [18] predicted evolution of meander using an enhanced 2D model and assuming a channel with constant width and a fixed computational domain. Yen and Ho [19] simulated bed evolution in channel bends with constant walls using numerical model. Molls and Chaudhry [20] proposed model for simulating the experimental data of flow in bend conducted by Rozovskii