9th International Congress on Civil Engineering, May 8-10, 2012
Isfahan University of Technology (IUT), Isfahan, Iran

# Numerical Study on Effect of Bend Radius on the Velocity Field in Curved Open Channels

**S. baghalian[1,2], H. Bonakdari[1,2] and M. Fazli[3]**

**1- Department of Civil Engineering, Razi University, Kermanshah, Iran**
**2- Water and Wastewater Research Center, Razi University, Kermanshah, Iran**
**3- Department of Civil Engineering, Bu-aliSina University, Hamedan, Iran**
Sara.baghalian@gmail.com

**Abstract**

In this study by 3D numerical simulation of flow field with ANSYS-CFX software, variations of velocity profiles in different sections of bend in three phases (i.e. air, water and sediment) were investigated. At first, the 90° bend with $R/B = 3$ was simulated and its results were verified using experimental data that have been obtained in an experimental analysis. Then by changing the ratio of bend radius to channel width, the effect of curvature radius on flow field in 90° bend was studied. Results showed that in $R/B < 3$ maximum of velocity mostly occurs in inner wall while in $R/B > 3$ secondary flow is stronger than of longitudinal flow, so this causes to transfer the maximum velocity from inner wall to outer wall.
**Keywords: Bend, CFD, ANSYS-CFX, Velocity filed, Bend radius.**

## 1. INTRODUCTION

Studying of flow patterns in curved channels is a subject of interest for researchers and engineers of the hydraulic science. Generally understanding flow patterns in curved channels is more complex than straight channels. Presence of turbulence in flow and complex structure of flow in bends causes phenomena such as deposit, erosion, changing in water level and etc. These phenomena increase importance of the study of flow in channels with bend. With advance of computers, the CFD is used for investigating hydraulic phenomena in complex cases such as curved channels and meandering rivers. The CFD using numerical techniques solves the Navier-Stockes equations for given geometry and boundary conditions. Also it solves the three-dimensional turbulent flow equations and utilizes a collocated and cell- centered storage scheme with a finite- volume discretization. In recent years some of researchers have studied the flow pattern in meandering rivers and curved channels. Jung and Yoon [1] by changing the discharge and the bed materials in an 180° bend analyzed the characteristics of flow and bed topography. Lien et al. [2] by considering affection of the secondary flow proposed a 2Dmodel for simulating flow field in channel bends. Blanckaert and Graf [3] and also Kassem and Chaudhry [4] studied the effect of secondary flow caused by centrifugal force on sedimentation and erosion in bend. Also De Marchis and Napoli [5] developed a 3Dnumerical model for investigating flow field in open channels with bend. Bonakdari et al. [6] using numerical and experimental investigations studied effect of a bend on the velocity field in a circular sewer with free surface flow. Also Bonakdari et al. [7] using numerical analysis, Artificial Neural Network (ANN) and Genetic Algorithm (GA) investigated and predicted the velocity field in curved open channels. Oh Baek and Won Seo [8] proposed a new equation to define the variations of the vertical profile of the transverse velocity along the centerline of curved channels. In another study Abhari et al. [9] using experimental and numerical study investigated flow pattern in a channel with a 90° bend. Also they used CFD software for numerical analysis.

In this study by 3D numerical simulation of flow field with ANSYS-CFX software, variations of velocity profiles in different sections of bend in three phases (i.e. air, water and sediment) have been investigated. First of all, the 90° bend with $R/B = 3$ was simulated and results of it was verified using experimental data that has been obtained in an experimental analysis in *TarbiatModares University of Tehran*. Then by changing the ratio of bend radius to channel width, the effect of curvature radius on flow field in 90° bend was investigated. Results show that in $R/B < 3$ maximum of velocity mostly occurs in inner wall while in $R/B > 3$ secondary flow is stronger than of longitudinal flow, so this causes transfer of maximum velocity from inner wall to outer wall.