10th International Congress on Civil Engineering, 5-7 May 2015
University of Tabriz, Tabriz, Iran

Comparison and numerical and experimental study of flow patterns in sharp bends

salma ajeel[1], Azadeh Gholami [2], Hossein Bonakdari [3,*], Ali Akbar akhtari [4,*], khosro morovati[1]

[1] M.Sc. Student, Department of Civil Engineering, Razi University, Kermanshah, Iran

[2] Ph.D. Student, Department of Civil Engineering, Razi University, Kermanshah, Iran.

[3]Associate Prof, Department of Civil Engineering, Razi University, Kermanshah, Iran.

[4] assistant Prof, Department of Civil Engineering, Razi University, Kermanshah, Iran.

[*] Water and Wastewater Research Center, Razi University, Kermanshah, Iran.

gholami.azadeh@yahoo.com

ABSTRACT

Study of flow pattern in sharp bends is so important because of the complexity of the flow pattern in this bends in to mild bends. In this study, the numerical study of flow pattern in three sharp bends 90°, 120° and 180° have been done. Corresponding numerical model is based on the Fluent software with free surface model. And the $k-\varepsilon$ (RNG) turbulence model is used to simulate turbulent flow parameters. The results show that the overall pattern of velocity distribution for the three bends have been studied are identical. In this bends, maximum velocity until around the end of the bend is located in the inner wall and is transmitted to the outer wall in the sections located after the bend. Study area tends to flow separation in the inner wall in the ending section of the bend indicates that by increasing the angle of curvature of the bend, bending effect on the tendency to flow separation within the channel width is increase. The shear stress distribution pattern at each three bend is identical and similar to the velocity pattern.

Keywords: sharp bends, numerical model, velocity distribution, flow separation, shear stress distribution.

1. Introduction

Various factors are involved on changing the bend flow pattern, distribution of velocity components and also the formation of secondary currents that their study will help to understand the behavior of the flow in channels and natural rivers bend. One of these factors is the radius of curvature of the channel (R_c), which is a parameter affecting the flow pattern of the bend. With the flow arrival to the bend, transverse gradient at the surface caused by the centrifugal force and in the channel bed, pressure gradient overcomes to the centrifugal force and in the water level, centrifugal force overcomes the pressure gradient that this process leads to the creation of swirling flow in the channel cross section, the current rotation is called the secondary flow. In mild bends, more opportunities for interaction between centrifugal forces and there pressure gradient and flow path long way through the length of the bend While in the sharp bends, the flow affected by the bend at once and so the flow behavior in this bends to the mild bends are more complicated. The study of