



Numerical Analysis of Flow Pattern around square Bridge Piers by New Ansys Software

Mohammad Vaghef¹, Hamed Dashtpeyma², Arash Adib³, Javad Roohian⁴

1- Assistant Professor, Department of Civil Engineering, Persian Gulf University, Iran.

2- Under graduate student, Department of Civil Engineering, Persian Gulf University, Iran.

3- Assistant Professor, Department of Civil Engineering, Shahid Chamran University, Ahvaz, Iran.

4- Under graduate student, Department of Mechanic Engineering, Persian Gulf University, Iran.

vaghefi52@gmail.com

Abstract

Bridges are one of the most important structures that their stability threatens by instability of their piers. The bridge collapse has been associated with scouring action of the flow around its piers. This complex phenomenon resulting from turbulent and vortex flow around piers.

For proper design of piers, scouring effect must be mentioned for stability of them. and scouring cannot be investigated before studying of flow pattern .

This paper is to report numerical analysis of flow pattern around square piers in rigid bed flowing water. Numerical analysis is based on software simulation with new generation of Ansys-CFX (12th Edition). But initial modeling will be performed with another software.

Important variation and factor that this investigation based on them are Froude number of flow.

At the end generated data from software simulation will be compared with laboratory researches. The locations of vortex systems and patterns of trailing wake-vortex systems will be found.

Keywords: Bridge piers, Flow pattern, Vortex flow.

1. INTRODUCTION

It's believed that scouring around bridge piers is one of the most important factors effecting on bridges stability. Because of the importance of this complex phenomenon many researchers have put their effort on experimental or numerical analysis of flow pattern [1, 2]. Some experimental analysis that conducted in this field is Vincenzo C.Santoro and Cowor Kers that they focused their study on effect of attack angle and velocity magnitude on maximum depth of scour. Another experimental study is Amed Rajaratnum (1998 & 2000) that their main aim of investigation concentrated on creation and improvement of downstream flow in front of piers due to flow direction.

Complexities of described condition have made numerical analysis a one of the best way for assessing of flow pattern and its effects [1, 4]. Some software simulation has been performed such as Richardson, Pencheng (1998) and Salahedin (2004) with using of Flow 3D and Fluent. Some researchers that their aim was study of shear stress around piers in bed are Kamil H.M.Ali and Othman Karim (2000) which their simulating software was Fluent (ver 4.3) with RNG and K-Epsilon turbulent model which their main defect was that Fluent (ver 4.3) couldn't simulate free surface models.

The main aim of this this article is numerical analysis of flow pattern around square pier based on software simulating using Ansys-CFX (ver 12.1 2010). Effecting variables that this simulation is based on are Froude numbers of flow in subcritical condition. Condition and specification of free surface and open channel has been mentioned directly.

In description of this phenomenon it's suitable to express the proses of occurrence in brief. When water flow approaches obstructed object it has to shape the geometry of it this made flow to change its normal behavior. When flow get close to vertical pier because of stagnation point that occurred in front side of pier a fraction of flow made to change its direction to down called downstream flow. This stream flow can be destructive and reason of scouring and horseshoe effect furthermore makes vortexes near bed that extends shear stress in lateral section of channel in pier location. It's economical that control the destructive flow with proper design of dimensions' of piers related to width of channel [3].