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 Vincenza 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 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].
2. **SOFTWARE SIMULATING LOGIC**

Transport equation that this simulation is based on:

\[
\frac{\partial (\rho \varphi)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \varphi) = \nabla \cdot (\rho \mathbf{D}_\varphi \nabla \varphi) + S_\varphi
\]  

Which \( \rho \) is the mixture density, mass per unit volume, \( \varphi \) is the mixture density, mass per unit volume, \( \varphi = \Phi / \rho \) is the conserved quantity per unit mass, \( S_\varphi \) is a volumetric source term, with units of conserved quantity per unit volume per unit time, \( D_\varphi \) is the kinematic diffusivity for the scalar (Ansys-CFX ver 12 Theory Guide).

For turbulent flow, this equation is Reynolds-averaged [6]:

\[
\frac{\partial (\rho \mathbf{U} \varphi)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U} \varphi) = \nabla \cdot (\rho \mathbf{D}_\varphi \nabla \varphi) + \mathbf{S}_H + \mathbf{S}_T
\]  

Which \( S_T \) is the turbulence Schmidt number, \( \mu_T \) is the turbulence viscosity (Ansys-CFX ver 12 Theory Guide).

Turbulent function for K-epsilon and rough surface expresses in term of turbulent viscosity [6].

\[
\frac{\partial (\rho \mathbf{U} \varphi)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U} \varphi) = \nabla \cdot (\rho \mathbf{D}_\varphi \nabla \varphi) + \mathbf{S}_H + \mathbf{S}_T
\]

Where \( S_H \) is the sum of the body forces, and \( \mu_{eff} \) is the Effective Viscosity defined by:

\[
\mu_{eff} = \mu + \mu_T
\]

and \( \mu' \) is a modified pressure (Ansys-CFX ver 12 Theory Guide) [6].

\[
\rho' = \rho + \frac{2}{3} \rho k_{C} + \frac{2}{3} \mu_{eff} \nabla \cdot \mathbf{U}
\]

3. **MODELING**

Initial conditions and parameters which performed in simulating considered by 4 different Froude number (0.1, 0.14, 0.2, 0.3) and constant rate of flow (25 Kg/s). The geometry of simulated flume has been gotten from common laboratory equipment.

Conditions that used for modeling is steady state for simulation type, K-epsilon for turbulent model, multiphase and Intensity and Length Scale for turbulent type. 2 steps of refinement adjusted for more accurate result and lower band of convergent set to 1e-4 (Figure 3). Refinement factor set to water volume fraction for better result in free surface and pier. Geometry and mesh generated by Ansys-Workbench components (Figure 4). The width of Flume is 0.6m and free surface level varies due to Froude number (Figure 1).

![Figure 1-Geometry of simulation (Left- 3D view) (Right- Top view)](Image)
3. **RESULTS**

The results show that as Froude number rises area with maximum velocity get extended near bed and pier. Figure 5 describes that maximum velocity occur in a corner near pier and bed. Center of vortexes get closed to bed as Froude number rises (Figure 6) this can be the reason of higher shear stress in lateral area of pier. For smaller Froude number shear stress spread normally in bed (Figure 7).
In figure 8 it can be believed that in front of pier because of obstructing object flow has to shape the pier and with gravity force a fraction of flow’s direction becomes downward and intensity of vectors’ slope get larger as flow go downward until it approaches bed (Figure 8). This fact can be the reason of periodic vortexes after pier and first reason of scour pit in front side related to normal flow direction. And downstream flow can make intense vortex in scour pit in front side. And tail vortexes can make horseshoe erosion after pier.
Figure 8 illustrates that the larger the Froude number, closer vortex occurs near bed with stronger strength. Downstream flow develops with sharper angle as normal velocity rises in magnitude. Flow direction in cross section tends to be downward and maximum volume of flow pass in smaller area close to bed. It can be perceived that critical conditions tend to occur near to bed such as maximum shear stress, the sharpest angle of downstream flow and the strongest vortex.
4. CONCLUSION

One of the most important factors in scouring study is investigation of stream flow near piers. Importance of this phenomenon can be proved by illustrating of shear stress contour in bed. Controlling of first effect of shear flow in bed can prevent second effect of scour depth on extension of erosion. As described before numerical analysis of flow pattern can stifle complex experimental laboratory works and results can be retrieve in fewer time and less laboratory equipment. From simulated condition it can be believed that as Froude number of flow get larger, maximum velocity focuses near bed and shear stress get on its peak in lateral sides of square pier. Final profile of deformed bed get influence from second effect of stream flow in deformed bed. It can be controlled by reduction of vortexes strength near bed. Proper design of piers considering the ratio of dimensions between width of channel and piers can reduce or stifle unwanted flow behavior.

11. REFERENCES

2. Vincenza.Sanoro, “velocity profiles and scour depth measurements around bridge piers”.
6. Ansys-CFX solver theory guide.