EPH - International Journal of Science And Engineering
ISSN (Online): 2454 2016
Volume 02 Issue 01 February 2016

DOI: https://doi.org/10.53555/eijse.v2i1.121

NUMERICAL METHOD FOR ESTIMATING FLOW PATTERN AROUND CONSECUTIVE HOCKEY-SHAPED SPUR DIKE WITH USING COMPUTATIONAL FLUID DYNAMIC

Ali Hooshmand Aini[1]*
*[1]Department of Civil Engineering, Ayandegan Institute of Higher Education, Tonekabon, Iran

Corresponding author:
Email: Ali_Hooshmand1983@yahoo.com +989112376114

Abstract:-
Using the Dike is an indirect method for controlling coastline erosion based on the modification of the flow pattern. The construction of this structure is with the aim of diverting the flow from erodible River coastline or establishing the appropriate route for directing the flow and sometimes for required depth of shipping. This paper uses a numerical model to study flow patterns around the Consecutive Hockey-shaped Spur Dike. Firstly, the geometric model and meshing of the structure are generated in GAMBIT software and then the model is solved with the software FLUENT. In this paper, we have chosen k-ε as turbulence model. The results of the analysis are presented as the flow pattern in the dike and the distribution of velocity counter in X and Y directions.

Keywords:- *Consecutive Hockey-shaped spur Dike, CFD, Flow Pattern, FLUENT, k-ε.*

I. INTRODUCTION

River dikes are considered as a major structural organization of the river. The dikes are crossstructures or transverse-structures that are constructed cross the natural river wall with the appropriate length and the proper angle to the general direction of flow; which cause the variation during the river flooding on the side and on the critical areas; they also guide the river flow to the central axis. With flow deviation, the river edges -in the contour of the existing natural wall with border line- gradually are to develop with the deposition of sediments and are to stable with the gradual establishment of vegetation cover in the longterm period. Therefore, over the time, the flow focus is in the middle section of the river range. The dikes a single structure or a series of consecutive structure are built on one or both sides of the river. Considering the functional goals, the length of the dikes may be very short up to long. The dikes with various shapes and features are made at the horizon level. A variety of dike forms in accordance with the plan of the river are as follows: straight dike, straight bell mouth spur dike, T-shaped dike, L-shaped dike, and hook-shape dike. The straight dike is located with straight axis and proper angle to the wall or river flow and its mouth generally is for protection against the local scour in the circle form. L-shaped dike is more effective, the rate of local scour in the mouth is little and also the ability of deposition of sediment in the contour of the series dikes is more. The angle of the dike mouth is recommended to be less than 10 degrees. Selecting other dike forms such as T-shaped or hook-shape dike is depended on river conditions and engineering judgment.

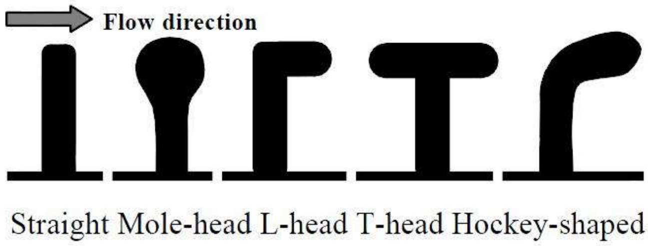

Fig 1. Types of Dike Shapes in Plan.

II. Mechanism of dike

The main role of impermeable dikes is the deviation of flow from the sides towards the middle of the river. The result of the flow diversion is the development of a rotating area of severe turbulence around the dike. This area appears in the distance between the dike upstream and downstream. In this area, a series of horizontal and vertical vortex are active, which create extreme turbulence in the surface of the water. The hydraulic process of this flow is the development of local scour area around spur dike, the deposition of sediments in the form of a long time deposition on the dike downstream and the river side and the substrate deformation in downstream of the river. While scouring is a local and serious threat to stability, durability and functionality of dike structures, the phenomenon of sedimentation along river downstream leads to the development and stabilization of the natural river wall. Rotating flow structure around the dikes depends on -in addition to the length and distance of dikes- the dike type, the shape of cross-section, side slopes along the dike and dike mouth, the position of the dike (in a direct line, internal or external wall), the angular arrangement of the dike towards the wall and the flow direction.

III. Flow pattern and bed topography around a single spur dike

The flow field around a single dike is different from the field around the dike series. In the dike operating conditions, the width of the river and the flow cross-section has been limited which causes the change in the structure of the flow movement in the scope of dike. By the upstream flow deviation to the middle of the River, the average speed and the width unit of main stream increase. The average speed increase leads to increase in the velocity gradient and the development of flow vortices and turbulence intensity around the dike. Figure 2 displays a simplified representation of the physical settlement of a dike in a rectangular channel with a flat and stable bed. Rotational flow field within the dike with trapezoidal cross-section is shown in Figure 3.

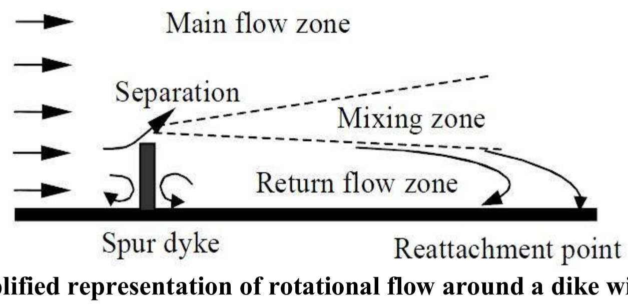

Fig 2. A simplified representation of rotational flow around a dike with a flat bed.

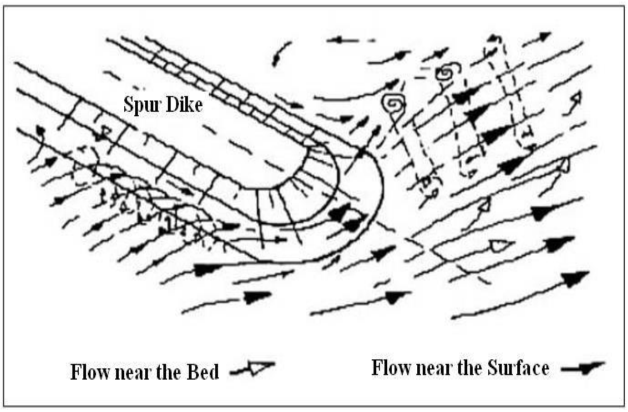

Fig 3. Structure of rotational flow around a dike.

IV. Numerical modeling

By analysis and design of hydraulic structures, assurance and more efficiency depends on the use of higher-performance methods and a deeper reflection of the flow behavior. The solving of the flow equations based on the numerical methods makes the determination of the flow parameters in a wide range possible. In general, these equations are the Navier - Stokes equations. Solving of these equations due to the non-linear nature of the equations contains particular complexity and by solving them we always encounter some problems.

In the past few years, solving of these equations was not possible because of the need of a large computer memory. Fortunately, today it is possible to solve the equations for many modes. In this regard, the specialized software which is designed for the flows can be used for time saving in addition to costs savings of the laboratory. The numerical methods used in CFD include limited elements method, limited volume method, limited difference methods and spectral methods. Among these methods, the method of FVM (limited volume method) can be used more, especially in the modeling of the non-dense flow. Most commercial software in the field of computational fluid dynamic have been developed and expanded based on this approach. To build the model, we have used the FLUENT software. FLUENT software is the pinnacle of programming to model fluid flow and heat transfer in complex geometry. This software has the capability of doing calculations with ordinary precision and double-precision standard and the user can select any of them as an option. This software is based on a limited volume method which is an appropriate method in computational fluid dynamic methods. For the numerical solving of the model, first we build geometry model, meshing and boundary conditions of the structures using GAMBIT program.

For simulation Flow around Consecutive Hockey-shaped Spur Dike, we assumed 3 spur dikes and according to the results obtained from Peterson (1987) and Richardson et all (1975) research, the distance between spur dikes are twice the effective length of spur dike (Figure4). As we can see we use velocity inlet and pressure outlet to solve problem.

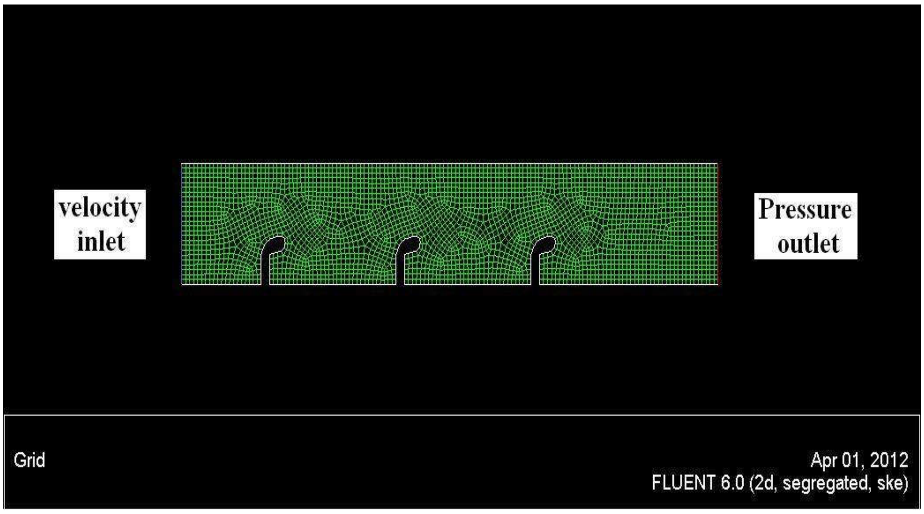

Fig 4. Geometric Models, Meshing and Boundary Conditions in the Model

Then, we transfer the geometry data to the FLUENT software. In the next step, we have to select the problem-solving method. FLUENT provides three ways to solve the problem: Segregated, Coupled Implicit and Coupled Explicit. These three formulations offer very precise results in a wide range of fluid types. In some cases, one of the formulations may be better than the others. As an example, by applying one of the methods the solving may be faster Convergent. Segregated method preferably can be applied to the non-dense or gently dense flow. But the Coupled method has been specifically developed for the high speed dense flow. [5] In this study, we used the segregated method for solving the flow. In this paper, we chose the k-ε as turbulence model. [6] K-ε model is a relatively complete and the general model but too expensive that is used to describe the turbulence properties by the average flow and influence. It is also useful for generation and sublation of turbulence. In this model, there are two transfer equations (partial differential equations PDE), one for the turbulent kinetic energy k and the other for the turbulent kinetic energy ε sublation rates. In the standard k-ε model, transfer equations (1) and (2) are applied that can be used for ε and k in the Fluent software: [8]

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\alpha+\frac{\alpha_t}{\sigma_k}\right)\frac{\partial}{\partial x_j}\right]+G_k+G_b \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_i}\left[\left(\alpha+\frac{\alpha_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2}$$

k = kinetic energy (per unit mass) of the turbulence
tα = turbulent viscosity
The equations include five adjustable constants, their values are as follow:

$$\sigma_k = 1.0 \quad C_\alpha = 0.09 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92 \quad \sigma_\varepsilon = 1.30$$

Gk= Terms of kinetic turbulent energy production is normal due to the average velocity gradient.
Gb = Terms of kinetic turbulent energy production is due to the Buoyancy force [8].
Solving the flow lines, flow velocity contours in the direction of X and Y, turbulence kinetic energy contours and flow pattern has been shown in Figures 5 to 8.

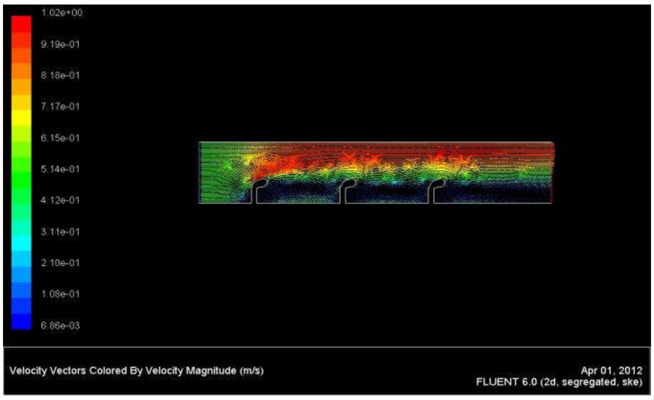

Fig 5. The velocity contours around Consecutive Hockey-shaped Spur Dike.

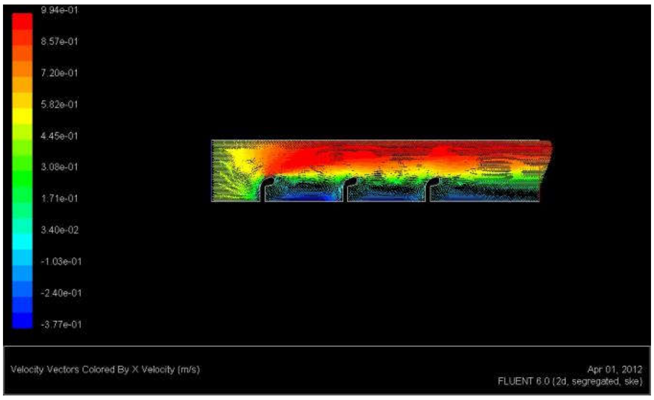

Fig 6. The velocity vectors in X direction around Consecutive Hockey-shaped Spur Dike

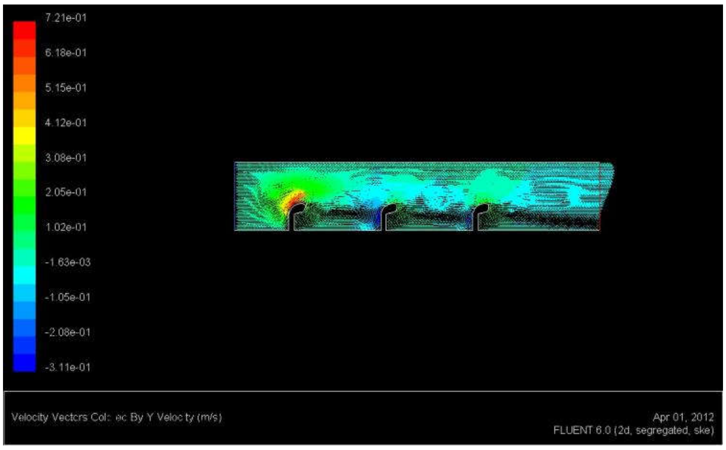

Fig 7. The velocity vectors in Y direction around Consecutive Hockey-shaped Spur Dike

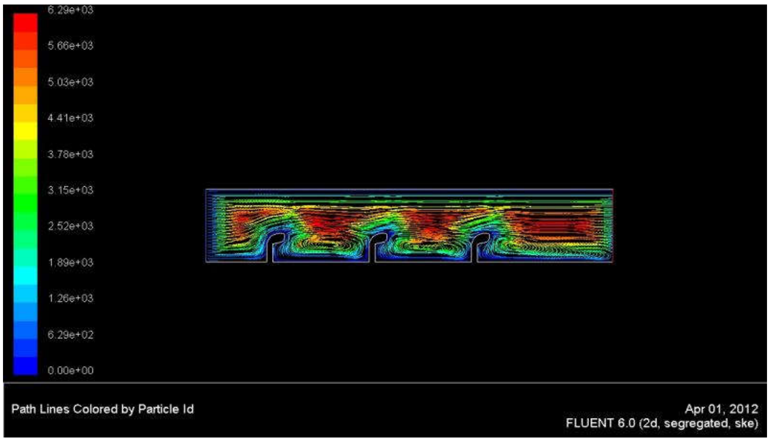

Fig 8. The flow pattern around Consecutive Hockey-shaped Spur Dike

V. Conclusion

According to the results of model, consecutive Hockey-shaped spur dike must be located in a way that flow pattern will not return to the beach and by put them at 2 times of the length of themselves the flow pattern don't strike to the beach (Figure-8). Between the spur dikes the velocity is always negligible (Figure-5). According to the flow pattern, maximum positive speed in the X direction at the top of the spur dike and maximum negative speed in the X direction in vortex flow along the wall has occurred (Figure-6). Maximum speed in the positive Y direction is occurred at the top of the first dike (Figure-7).

According to the values and the forms, it can be concluded that the software FLUENT has a high ability in modeling the flow around Hockey-shaped spur dike and by means of this software the flow parameters can be achieved in a satisfactory condition.

References

[1]. Azinfar, H., Flow Resistance and Associated Backwater Effect Due to Spur Dikes in open Channel," PhD Thesis, Department of Civil and Geological Engineering University of Saskatchewan, winter 2010.

[2]. Kuhnle, R., Jia, Y and Alonso, C., Measured and Simulated Flow near Spur Dikes," us China Workshop On Advanced Computational Modeling in Hydro Science & Engineering, Oxford, Mississippi, USA, 2001.

[3]. Naeini, S., Effect the length and distance between spur Dike on flow around it, 7th conference of hydraulic, University Tehran, 2008.

[4]. Eshaghiye, P., Talebbidokhti, N.,Numerical Investigation of Spur Dike with using VOF, 4th conference of civil Engineering, university Tehran, 2009.

[5]. Soltani M., Rahimi Asl R., Computational Fluid Dynamics using Fluent, Tarrah, 2007, p.p 84-92.

[6]. Shojaie Fard M. and Noorpour Hashtroudi A., Introduction to Computational Fluid Dynamics, Iranian University of Science and Technology, 2007, p.p 131 -139.