J. Civil Eng. Mater.App. 2017 (September); 1 (2): 33-38

Journal of Civil Engineering and Materials Application

Journal home page: http://journals.lexispublisher.com/jcema

Received: 10 April 2017 • Accepted: 25 June 2017

Research

doi: 10.15412/J.JCEMA.12010201

Investigating LES Turbulence Model in Modeling the Velocity Distribution Around the Hydraulic Structure Using ANSYS

Ahmad Safari[1*], Behrooz Moradi Mofrad[1], Sayed Hamid Reza Barnjani[2]

[1] Department of Civil Engineering, Yasuj Branch, Yasouj University, Kohgiluyeh and Boyer Ahmad, Iran

[2] Department of Civil Engineering, Bushehr Unit, Islamic Azad University, Bushehr, Iran

*Correspondence should be addressed to Ahmad Safari, Department of Civil Engineering, Yasuj Branch, Yasouj University, Kohgiluyeh and Boyer Ahmad, Iran; Tell: +989176262363; Fax: +987432228450; Email: ahmad.s0210@gmail.com.

ABSTRACT

Spur dikes are river training structures used for prevention of erosion at river banks and cause distancing of flow from the critical zone and creation of local contraction in water flow. In the present study, 2D modeling of the flow pattern around a spur dike in a straight canal with 6m length and 0.45m width is presented. Also the finite element method is utilized for solving the differential equations. Modeling of the turbulent flow around a single spur dike is performed using k-ε model in ANSYS software and the results were compared to those of the experimental study. The results show that the finite element method, by incorporating the k-ε model, models the flow pattern around the spur dike well and yields an average error value of 12.57%, and this shows a good agreement between the numerical modeling results and those of the experimental study.

Key words: Spur dike, Flow field, Finite element, LES turbulence model.

Journal of Civil Engineering and Materials Application is published by *Lexis Publisher*; Journal p-ISSN xxxx-xxxx; Journal e-ISSN 2588-2880.

1. INTRODUCTION

Spur dikes are the river structures used to prevent erosion of the river banks and cause distancing of flow from the critical zone and cause local contraction in the water flow (1). This structure affects the flow line and when the flow encounters the spur dike, it experiences circulation and complicated oscillations. Understanding the function of these rotational flows is essential for understanding the scouring phenomenon around the spur dikes (2). In this respect, various experimental and numerical studies have been performed. Nagata et al. (2005) have numerically simulated the 3D flow pattern around a single spur dike with live bed and, while analyzing the structure of mean flow, they investigated its relation with occurrence and development of scour hole. According to the results obtained by the abovementioned researchers, at the start of computations, the flowlines are concentrated at the spur dike tip. In the case of a balanced bed, due to formation of scour hole, the dimensions of rotational zone are expanded and flow lines are transferred to the canal center. In the case of the flat bed, the rotational flows are not adequately developed but by occurrence of scour hole, the rotational flows are intensified and while having intense downstream flow at the upstream side of the spur dike, the recirculation flow also is intensified at the vicinity of bed (3). Koken and Constantinescu (2008) carried out comprehensive studies on the structure of 3D currents (averaged and instantaneous) around a single spur dike in a straight canal in two cases of the fixed flat bed and deformed bed using the Large Eddy Simulation (LES) method. They have referred to occurrence of two types of horseshoe vortices; the first type is formed due to downstream flow at the upstream side of the spur dike and its mixture with the boundary layer of the incoming flow (4). Duan (2009) studied the mean and turbulent flow structures around a straight spur dike in a laboratory canal with fixed flat bed. She measured the flow field in 8 deep layers and at upstream and downstream of the single spur dike. According to her findings, the mean velocity is separated in both lateral and vertical directions and, at the back of the spur dike within the circulation zone, a combination of horizontal and vertical vortices is observed (5, 6). Nagy (2005) investigated the angle of vane-shaped dike with respect to the bank and Froude number in two submerged and non-submerged cases for a spur dike in a straight alignment. He found that the flow pattern for the submerged spur dike exhibited less turbulence with respect to the non-submerged one and the flow velocity around the non-submerged spur dike is more reduced with respect to that of the submerged spur-dike (7). Uijttewaal and Booji (2000, 2005), experimentally investigated the flow pattern around a trapezoidal shaped submerged spur dike with four side slopes in a straight alignment. The results showed that in permeable spur dikes, contrary to the impermeable ones, the momentum of flow over the spur dike body disrupts the rotational flow structure at the end of spur dike and in fact prevents formation of the rotational pattern (8, 9). Elawady et al. (2000), by examining the parameters such as the length and height of spur dike, flow rate, submergence percentage, and angle of the spur dike with respect to the bank, investigated the flow pattern for a vane-shaped dike in a straight alignment. They found that in case of high submergence upstream of the spur dike, at layers close to the bed, some vortices were formed perpendicular to the flow direction and toward upstream of the spur dike. At downstream of the spur dike they had the shedding state. Whereas in case of low submergence the layers close to the bed, flowed with a high velocity downward along the spur dike and no vortex was observed upstream of the spur-dike (10). Kumar and Malik (2016), investigated the flow pattern around a single repelling-type spur dike with angles of 45 and 60 degrees with respect to the flow axis , and also the T-shaped straight spur dike by FLUENT software and presented the flowlines and velocity distribution at two levels; one close to the water surface and other close to the bed. They concluded that the effect of spur dike shape is much more visible at the level which is close to the bed in comparison to the water surface level. But the effect of Froude number in all cases is negligible both in terms of the spur dike shape and angle with respect to the flow direction (11). Salamatian et al. (2016) experimentally studied the flow pattern and shear stress for a triple series of vane-shaped and straight dikes in a 90 degrees bend. They located the triple series in three positions of 30, 45 and 90 degrees and measured the flow field considering a range of 20 degrees higher and 20 degrees lower than these values using a 2D velocimeter. The results showed that, by movement of the triple series downward, the length of separation zone is reduced but its width remains constant (12). In the present research modeling of the turbulent flow pattern around a single spur dike is presented using the ANSYS software and application of the finite element method and LES turbulence model. Modeling is performed based on the experimental study by Asadzadeh (2012), and the obtained results of modeling are compared to the results of this research.

2. NUMERICAL MODELING

ANSYS software is among the analysis tools in which the finite element method is incorporated for modeling and analysis. The finite element method is developed for solution of complex problems with arbitrary geometries and varying material types, and loadings. In this method the complex models are first divided into smaller solvable elements. Then, by combining the results of solving each element, the total response of model is obtained for each point. The capability of analyzing various problems and enhancement of computer tools have caused widespread use of the finite element method and the software produced based on it (13).

3. GOVERNING EQUATIONS

For modeling of this problem the finite element method is used, and to account for the flow turbulence effects, the k-ε model is applied. The governing laws of an incompressible viscous fluid flow are stated by a continuity equation and three momentum equations along the three coordinate axes which are known as Navier-Stokes equations. These equations in fact state the consistency of mass and momentum in mathematical language. The continuity equation or the conservation of mass equation in a fluid flow is stated as equation (1) (14):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

Navier-Stokes equations are the governing momentum equations for a viscous, Newtonian fluid flow and the tensor form of these equations is stated in the Cartesian coordinates as equation (2):

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \quad (2)$$

The left hand side of the above equation states the fluid acceleration which includes the temporal and spatial changes of the velocity. The terms at the right hand side state the force per unit mass. The basic idea behind LES is that all turbulent structures which could be computed using a computational grid are directly computed and only very fine structures which could not be computed are approximated. When using LES, care must be exercised that this model is a 3D model and time dependent. Furthermore, the oscillations should be initialized or defined at input boundaries. In this model the turbulence effects that are very small and could not be computed, should be represented by the turbulence viscosity which is proportional to a length scale multiplied by a criterion of the oscillations in that scale. This model uses a geometric mean of the grid cell dimensions for the length scale (15):

$$L = (\delta x.\delta y.\delta z)^{\frac{1}{3}} \quad (3)$$

And it scales velocity oscillations by multiplication of L magnitude and the mean shear stress. These quantities are combined to present the kinematic eddy viscosity of the LES model:

$$\nu_T = (C.L)^2.\sqrt{e_{ij}e_{ij}} \quad (4)$$

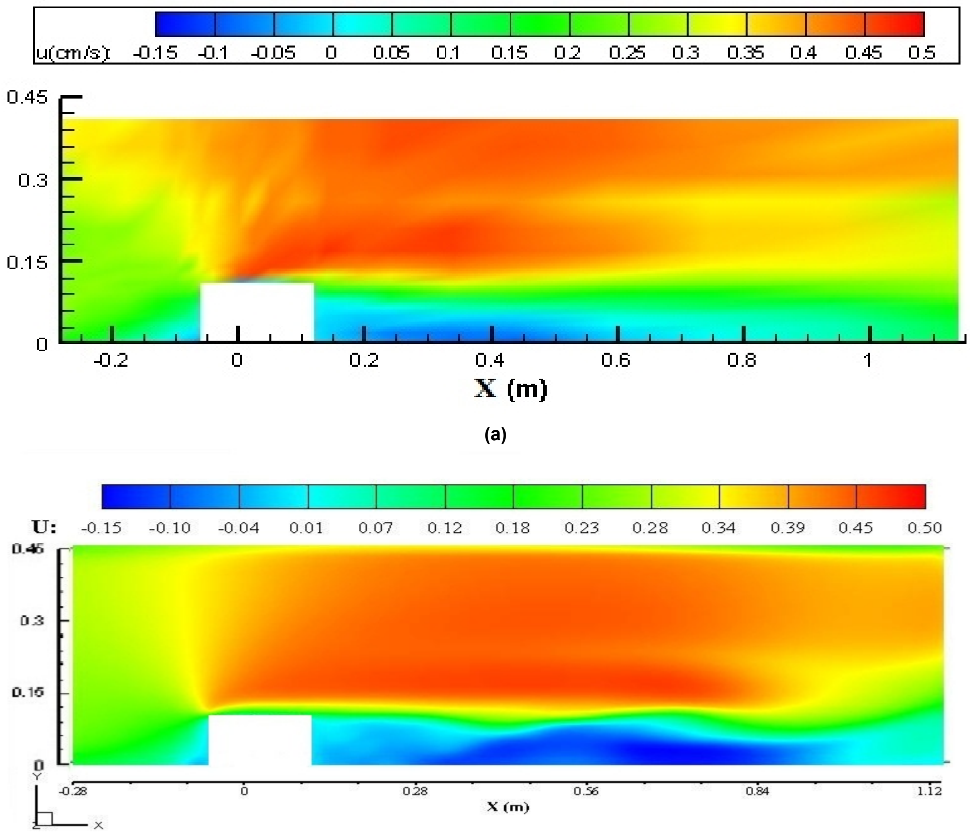
Where C is a constant coefficient ranging from 0.1 to 0.2 and $\boldsymbol{e_{ij}}$ is the rate-of-strain tensor.

4. MODEL PREPARATION

The experimental study of this research was performed in 2012 at Tarbiat Modares University in a canal with 6m length and 0.45 m width. The used spur dike in this experiment has a length of 10cm, width of 5cm and side slope of 75 degrees and the tip slope is zero. Also the mean flow velocity before the spur dike is 0.3m/s and the flow depth is 11cm. The crest center is located 2.5m away from the beginning of the canal and the flow field is measured 30 cm upward from the upstream spur dike and 1.14m downward from the crest center. Thus for modeling the problem , the intended geometry with a total length of 3.64m and width of 0.45m was built using ANSYS software. It was performed in a way that, first, according to the experimental study, the two dimensional figures of canal and spur-dike were drawn and gridding was performed with 1cm distance between lines. Then, the boundary conditions for velocity and pressure were applied at different surfaces so that the velocity at the input surface was taken 0.30m/s and velocity gradient at the output surface and wall surfaces was taken zero also the pressure at the output surface was taken zero and the pressure gradient at the input surface and wall surfaces was taken zero. Modeling of the turbulent flow in this solver was conducted by the LES model.

5. RESULTS AND DISCUSSION

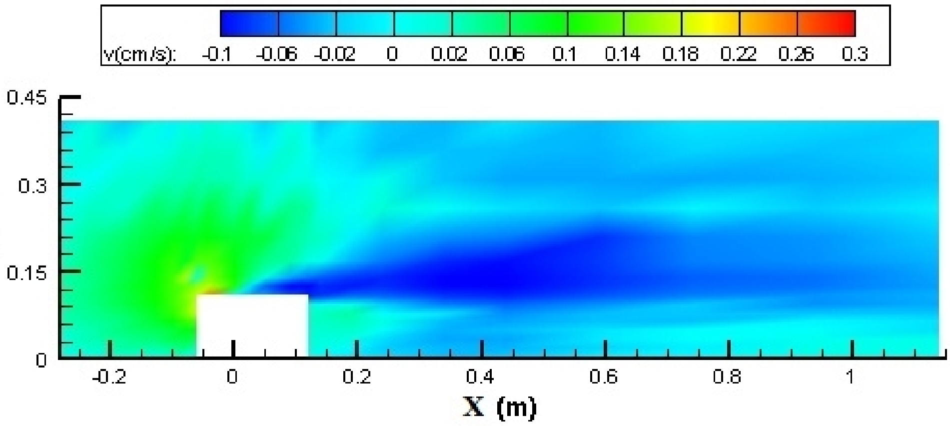
By moving toward the spur dike, the flow was affected by the spur dike and the flow lines were gradually diverted from the walls of the spur dike toward central part of the canal. The highest value of longitudinal velocity component was at the front of the spur dike which was due to contraction of the flow width. The highest value of negative longitudinal velocity and the highest value of negative transverse velocity occurred at the back of the spur dike and this indicates the occurrence of rotational flow at the back of the spur-dike. Constriction of flowlines at upstream edge of the spur dike is completely evident and transverse diversion of flow at upstream edge of the spur dike indicates the highest value of positive transverse velocity component. The results show that, in terms of the flow pattern, numerical modeling is compatible with the experimental research (Figure 1 and **Figure 2**).



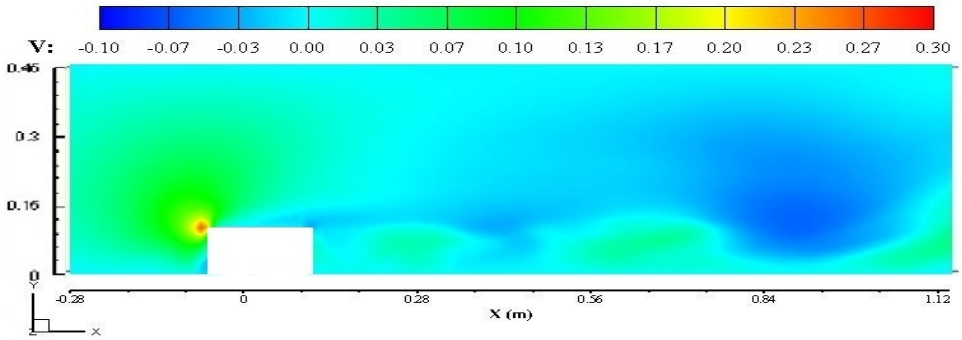
(a)

(b)

Figure 1. Velocity contour around a single spur dike for the longitudinal velocity component at the level of 2cm from the bed; a) Experimental study; b) Numerical modeling



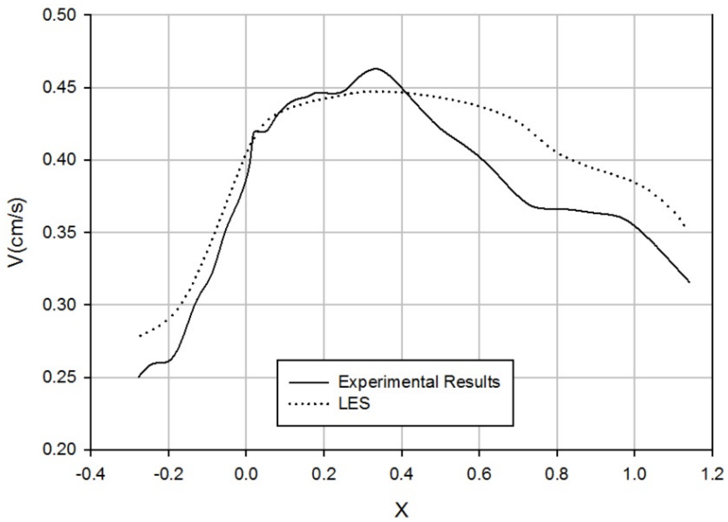
(a)



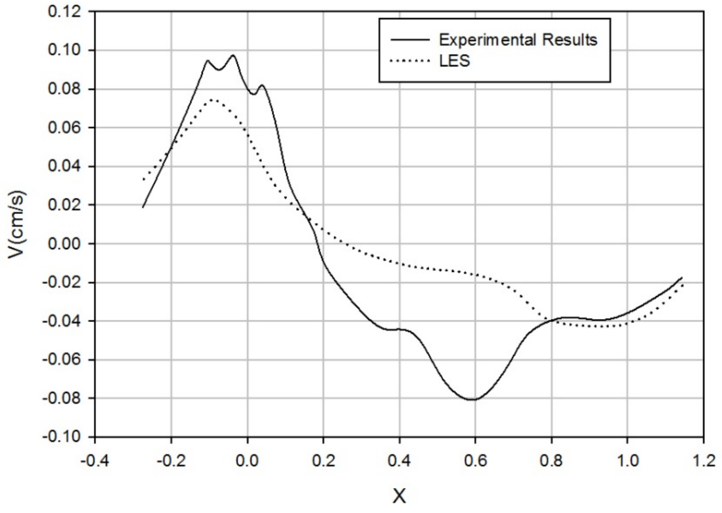
(b)

Figure 2. Velocity contour around a single spur dike for the transverse velocity component at the level of 2cm from the bed; a) Experimental study; b) Numerical modeling

For quantitative comparison of velocity values, variations in the longitudinal and transverse velocity components at the flow centerline are shown in Figure 3. The results indicate that, in addition to the flow pattern around the spur dike, the values of longitudinal and transverse velocity components along the flow centerline also exhibit a good compatibility.



(a)



(b)

Figure 3. Velocity values in the experimental and numerical models along the canal centerline, a) longitudinal velocity; b) transverse velocity

For quantitative investigation and comparison between the experimental results and the present model results, the error values are calculated by the following expression (16):

$$\text{Error} = \frac{1}{N}\sum_{i=1}^{N}\frac{\varphi_i - \varphi_i^{'}}{\varphi_i^{'}} \times 100 \quad (5)$$

In this equation, N is the number of data, $\emptyset_i$ is the quantity value obtained from the present model and $\emptyset_i^{'}$ is the quantity value from the experimental study. Using the experimental data and the results of numerical model (Figure 3), the corresponding error of simulating the longitudinal velocity component is 10.31% and that of the transverse velocity component is 14.83%. Thus the presented model has a mean error of 12.57% which is in the acceptable range.

6. CONCLUSION

In this research the modeling of the flow pattern and also distribution of longitudinal and transverse velocities around the spur dike were performed using ANSYS software. The results indicate that the maximum positive longitudinal velocity occurs at the front of the spur dike. Also the maximum positive transverse velocity occurs at the upstream edge of the spur dike. The maximum negative longitudinal and maximum negative transverse velocities occur at the back of the spur dike which in turn cause rotational flow. Also distribution of longitudinal and transverse velocity components exhibited good compatibility and the results obtained by comparison between values of longitudinal and transverse velocities at the canal centerline showed a mean error of 12.57% which is within the acceptable range.

FUNDING/SUPPORT

Not mentioned any Funding/Support by authors.

ACKNOWLEDGMENT

Not mentioned any acknowledgment by authors.

AUTHORS CONTRIBUTION

This work was carried out in collaboration among all authors.

CONFLICT OF INTEREST

The author (s) declared no potential conflicts of interests with respect to the authorship and/or publication of this paper.

REFERENCES

1. Osman AM, Thorne CR. Riverbank stability analysis. I: Theory. Journal of Hydraulic Engineering. 1988;114(2):134-50.
2. Lagasse PF. Riprap design criteria, recommended specifications, and quality control: Transportation Research Board; 2006.
3. Nagata N, Hosoda T, Nakato T, Muramoto Y. Three-dimensional numerical model for flow and bed deformation around river hydraulic structures. Journal of Hydraulic Engineering. 2005;131(12):1074-87.
4. Koken M, Constantinescu G. An investigation of the flow and scour mechanisms around isolated spur dikes in a shallow open channel: 1. Conditions corresponding to the initiation of the erosion and deposition process. Water Resources Research. 2008;44(8).
5. Duan JG. Mean flow and turbulence around a laboratory spur dike. Journal of Hydraulic Engineering. 2009;135(10):803-11.
6. Duan JG, He L, Fu X, Wang Q. Mean flow and turbulence around experimental spur dike. Advances in Water Resources. 2009;32(12):1717-25.
7. Nagy HM. Hydraulic evaluation of emerged and submerged spur-dikes: temporal bed evolution and equilibrium state characteristics. Alexandria engineering journal. 2005;44(2):279-90.
8. Uijttewaal W, Booij R. Effects of shallowness on the development of free-surface mixing layers. Physics of fluids. 2000;12(2):392-402.
9. Uijttewaal WS. Effects of groyne layout on the flow in groyne fields: Laboratory experiments. Journal of Hydraulic Engineering. 2005;131(9):782-91.
10. ELAWADY E, MICHIUE M, HINOKIDANI O. Experimental study of flow behavior around submerged spur-dike on rigid bed. PROCEEDINGS OF HYDRAULIC ENGINEERING. 2000;44:539-44.
11. Kumar M, Malik A. 3D Simulation of Flow around Different Types of Groyne Using ANSYS Fluent. Imperial Journal of Interdisciplinary Research. 2016;2(10).
12. Salamatian S, Forghani M, Tabarestani MK. Flow Pattern and Stress

Distribution around Three Spur Dike in Ninety Degree Bend. International Journal of Engineering and Technology. 2016;8(6).
13. Manual AUs. Ansys. Inc Modeling, CFX. 2000;11.
14. VAGHEFI M, SHAKERDARGAH M, AKBARI M. Numerical investigation of the effect of Froude number on flow pattern around a submerged T-shaped spur dike in a 90° bend. Turkish Journal of Engineering and Environmental Sciences. 2015;38(2):266-77.
15. Tang X, Ding X, Chen Z. Large eddy simulations of three-dimensional flows around a spur dike. Tsinghua Science & Technology. 2006;11(1):117-23.
16. Karami H, Basser H, Ardeshir A, Hosseini SH. Verification of numerical study of scour around spur dikes using experimental data. Water and environment journal. 2014;28(1):124-34.