Influence of Spur Dike on Flow Patterns in an Open Channel

USMAN GHANI*, SHAHID ALI**, AND SABAHAT ARIF***

RECEIVED ON 27.12.2012 ACCEPTED ON 20.03.2013

ABSTRACT

Spur dikes are used for river protection purposes. They are typical in-stream structures. There existence results in a considerable change in the flow structure of open channel flow both upstream and downstream of the spur dike. This paper presents a numerical work conducted to get the mean and turbulent flow features under the influence of spur dike. Primary velocity distributions over cross sections and horizontal planes, streamlines over vertical sections and turbulence kinetic energy were investigated. The presence of spur dike was found to have disturbed all the investigated flow features along the length of the channel. The flow separation and recirculation was observed on the down stream side of the dike. The reversal of flow behind the dike was directed downwards from the surface. On the basis of the results obtained in this study, an attempt has been made to enhance the understanding of the flow patterns which exist in case of a spur dike which can further be used for development and improvement of formulae relevant to spur-dikes.

Key Words: **Spur Dike, Impermeable, Flow Separation, Navior-Stokes Equations.**

1. INTRODUCTION

Spur dikes are the structures which extend from the channel banks and project into the flow. There primary function is to protect natural river channels from erosion. A spur dike is an effective structure for bank protection which is being used world wide with confidence. The dikes redirect flow and trap suspended sediments in back water zones. Dikes also result in the formation of a safe pool for natural habitats. There are two types of dikes, first one is permeable and the second one is impermeable dike. The permeable dikes normally consist of several rows of reinforced concrete, steel or timber piles while the impermeable dikes are built with stones, rocks, gravel and soil. Permeable dikes are also termed as pile dikes. The construction of a dike results in a considerable change in flow characteristics both upstream and downstream of the dike. It results in flow separation on the downstream side of the channel. The detailed experimental as well as numerical study of a spur dike needs basic knowledge of hydraulics and CFD (Computational Fluid Dynamics) which is available in a number of standard books [1-3].

---

* Assistant Professor, Department of Civil Engineering, University of Engineering & Technology, Taxila.
** Senior Engineer, Atomic Energy Commission, Islamabad Pakistan.
*** Associate Professor, Department of Architecture Engineering & Design, University of Engineering & Technology, Lahore.

Although a lot of work has been done on flow behavior around a bridge pier but less focus has been paid to spur dikes so far. Recently Jennefier [4] did experimental work on fixed flat bed spur-dike in an open channel. She tried to understand the flow characteristics around the dike by measuring velocities with the help of an ADV (Acoustic Doppler Velocity) meter. Similarly Ahmad, et. al. [5] conducted research on flows behavior around a wall abutment. Some researchers did work on permeable dikes [6-9] in the past. Some of these studies included both flow and scour behavior around the spur-dikes or similar structures. Research has also been done with the help of numerical tool to understand flow characteristics of a spur-dike. For example, Mayere, et. al. [10] developed a numerical model for this purpose and first verified it and then used it to get different flow behavior. Tang [11] utilized numerical technique to investigate secondary flow and sediment deposition pattern in the presence of a spur-dike. Kimura, et. al. [12] used a non-linear two equation turbulence model for flow study around a bluff body whereas Ho, et. al. [13] modeled flow in the vicinity of a groyne. Despite all the above efforts, there is still much need for a comprehensive understanding of flow in the presence of a dike so that the existing formulae can be improved in the light of this enhanced understanding.

In the present work, a numerical model has been used for simulating the flow behavior around a spur dike and for enhancing the understanding of flow due to the spur dike. The spur dike was non-submerged. A 3D (Three Dimensional) CFD code FLUENT 12 [14] has been used for this purpose. The model was first validated using available data from literature. The numerical xperiments were then performed for a flat bed dike. Primary velocities, stream lines representing the secondary flow field, and turbulent kinetic energy were investigated on various longitudinal and transverse sections both upstream and down stream of the spur dikes. The results were analyzed and discussed to improve the knowledge regarding spur-dikes.

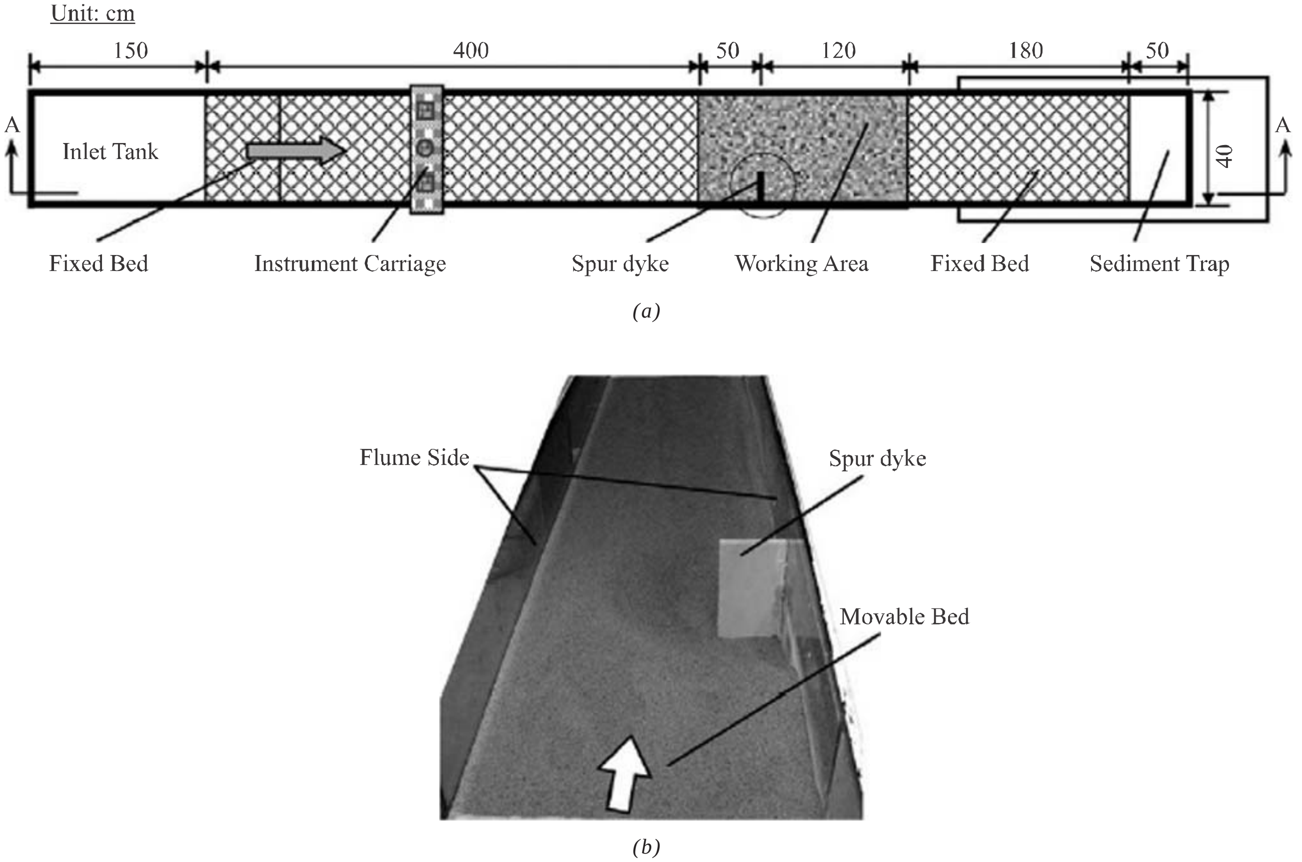
2. NUMERICAL MODEL FOR SPUR DIKES

The fundamental governing equations for all three dimensional numerical codes are Navior-Stokes equations. These are 3D continuity and momentum equations. In the present work, first of all the published data from literature was used for validating the numerical model of spur dikes and then numerical experiments were conducted to get different flow features. The validation is an important aspect of any type of numerical simulation work because it shows the ability of the model to handle the problems under consideration and it is achieved once the modeled results match the experimental data. For the present work the data of Zhang, et. al. [15] has been used for validation. They performed experimentation in an 8 m long channel. It had a cross-sectional dimension of 40x40cm. At the upstream, there was a 1.5m long inlet tank. The spur dike was located at a distance of 4.5m down stream the inlet. It had a thickness of 1cm and projected perpendicularly into the channel with a length of 10cm. It was an impermeable spur dike with painted wooden plate material. The channel had a slope of 0.001. The Reynolds number and Froude's number for this flow case were 14,250 and 0.41 respectively. The mean velocity of the water was 0.29 m/sec. The experimental channel has been shown in Fig. 1(a-b).

A numerical work is comprised of three steps i.e. pre-processor, solver and post-processor. The pre-processor used in this work is GAMBIT 6.3. It is used for creation of geometry, meshing the geometry and for assigning the boundary conditions to different surfaces of geometry. In this way, GAMBIT 6.3 has been used for pre-processing the spur dike problem. The mesh was comprised of unstructured triangular elements. The Fig. 2 shows the plan view of the mesh. A fine grid has been used close to the spur dike while the density was gradually reduced as the distance from the dike increased. The mesh independence test showed that the results obtained from the present mesh will remain almost unaffected from any further refinement of the mesh. This employs that our results were mesh independent. For this purpose, three different meshes were tested and results have been shown in Fig. 3.

FLUENT 12.0 has been used as a solver for the present research work. Different options are to be chosen for any type of numerical modeling keeping in view the problem under consideration. In this case, the following selections were made. The k-ε turbulence model was selected with SIMPLE algorithm. The second order upwind scheme has been used. The boundary conditions were given at the walls, bed, entry of the channel, channel exit and at the free surface. As we know the velocity values are zero at the walls and there is zero slippage between the wall and fluid particles, so a no slip boundary condition was assumed at the walls. The free surface treatment was achieved by assuming a rigid lid boundary condition. The velocity inlet and pressure outlet boundary conditions were assigned at the entrance and exit of the computational domain.

As turbulence model will not be used in the regions close to the walls. Instead, it has been accomplished through a



(a)

(b)

FIG. 1. PLAN VIEW AND THREE DIMENSIONAL VIEW OF EXPERIMENTAL SET-UP [15]

standard wall function. The wall function covers the regions close to bed and modeling is done only in the turbulent flow regions. The convergence criteria was set as 1x10^{-6}. The simulated surface velocity results matched the experimental values, so the model can be used for further research work of spur-dikes. After validation, the numerical experiments were done for a flat bed dike and results have been discussed below.

3. RESULTS AND DISCUSSION

3.1 Primary Velocity Contours

Fig. 4(a-g) shows the distribution of primary velocities (mean streamwise velocities) over sections along the channel. The primary velocity contours have been shown over three sections upstream the dike (at 4, 4.3 and 4.45m) while over four sections downstream the spur-dike (4.55, 5, 5.5 and 6m). The velocity contours are indicating the separation of flow and recirculation processes downstream the dike.

The velocity values are positive and increasing in Fig. 4(a-b) while these reduce suddenly over section 4.45 (just upstream of the dike). The velocity distribution immediately downstream the dike (at section 4.55m) indicates that on the left region, velocity values have turned negative that is a reversal of flow has occurred in this region. This results in flow separation and possible erosion of bed downstream the dike. As the distance from the dike increases (Fig.4(e-g)), this negative velocity intensity keep on decreasing, till the velocities over the entire section again become similar to the one which were noticed upstream the dike. The Fig. 5 indicates the velocity profiles at two different locations along the depth of channel.

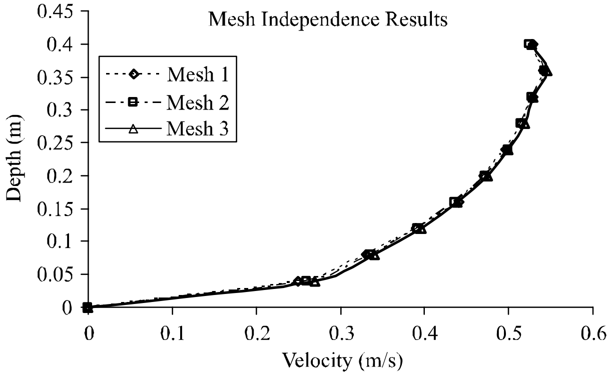


FIG. 3. MESH INDEPENDENCE TEST RESULTS SHOWING VELOCITY PROFILES FOR THREE DIFFERENT MESHES

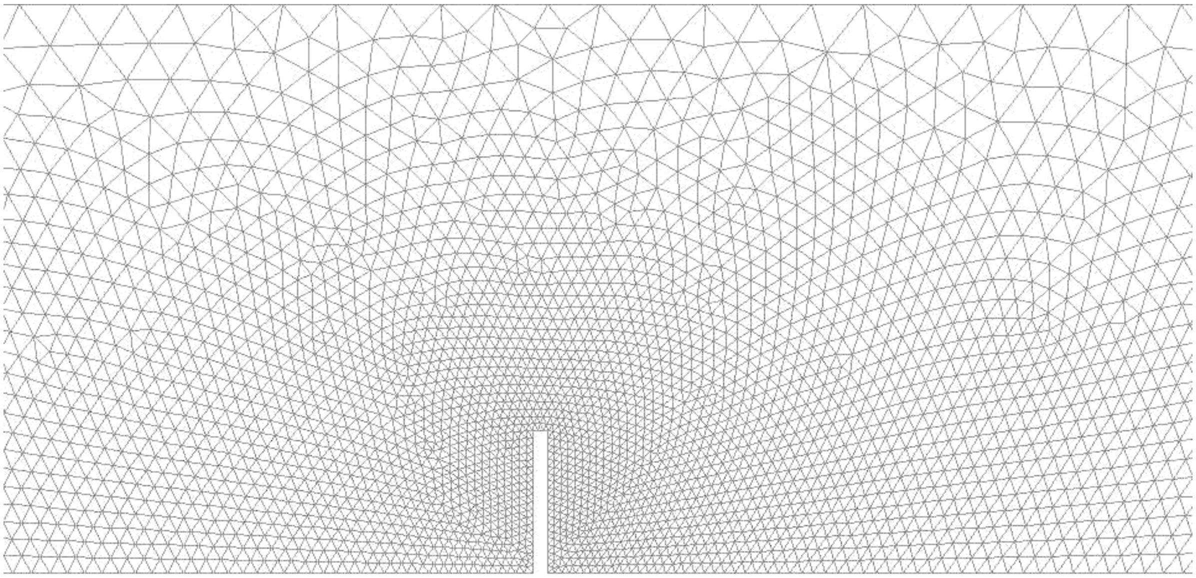


FIG. 2. PLAN VIEW OF THE MESH USED IN THIS WORK

3.2 Streamlines Over Longitudinal Vertical Sections

The stramlines have been plotted in Fig. 6(a-b) over vertical longitudinal sections passing perpendicular to the spur-dike at a lateral distance of 0.05 and 0.07 m from the bank of dike. The diagrams have indicated the flow reversal which continues upto around five times the length of the dike and which contributes to scouring processes.

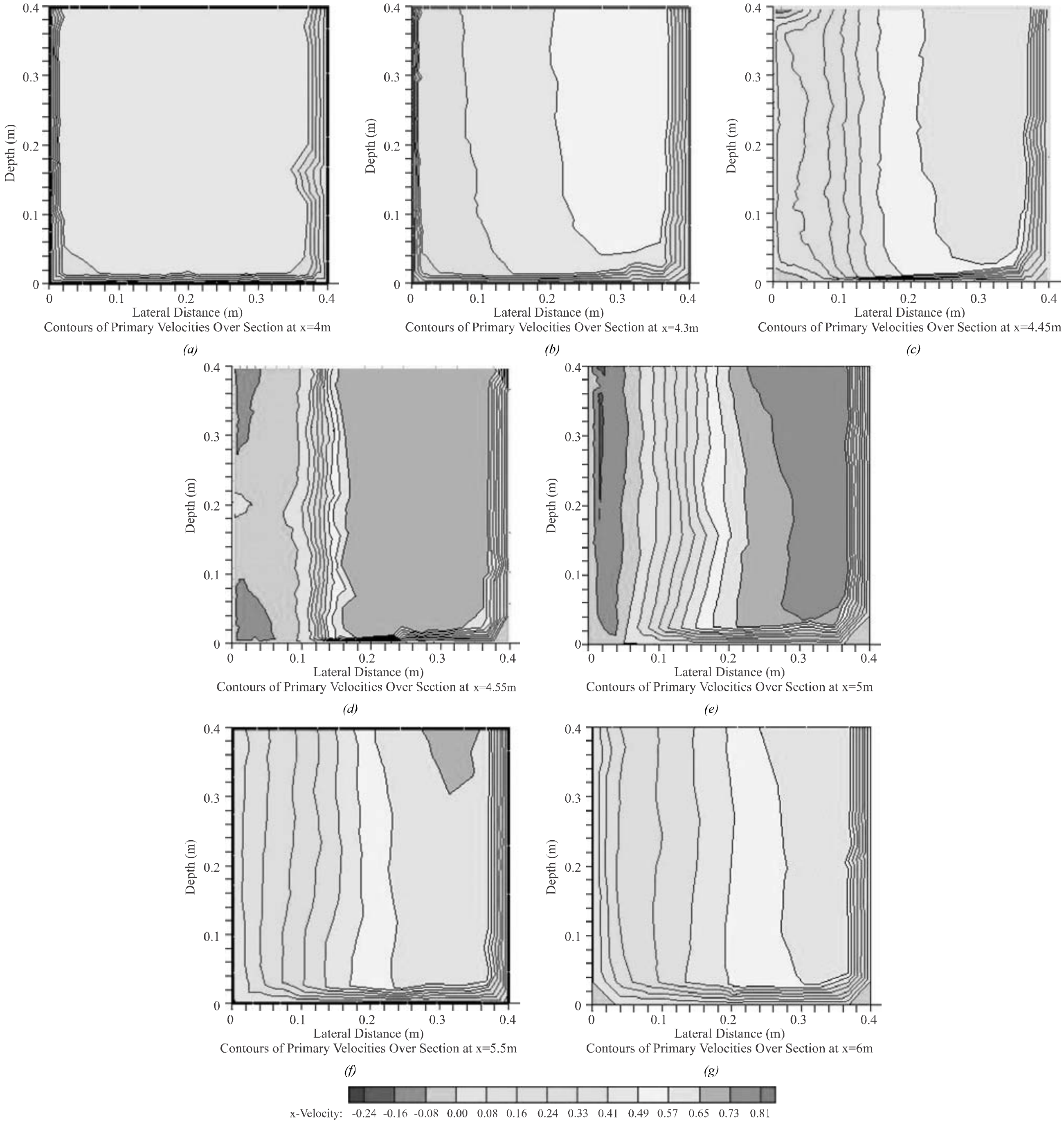


FIG. 4(a-g). MEAN VELOCITY DISTRIBUTIONS OVER DIFFERENT CROSS-SECTIONS ALONG THE CHANNEL

3.3 Velocity in a Horizontal Plane

The Fig. 7 shows the velocity distribution over a plane parallel to bed at a height of 0.27m. It clearly shows the reversal and circulation of velocity behind the spur dike. From the upstream side (from left to right) the flow is more on the dike regions but behind it, it has a separation behavior. This is an indication that flow has changed its direction immediately after passing through the dike and a reversal of flow has happened. The influence of dike on the flow features is less in other parts of the domain. This negative flow reduces in magnitude as the distance from the dike increases both in stream-wise and lateral direction.

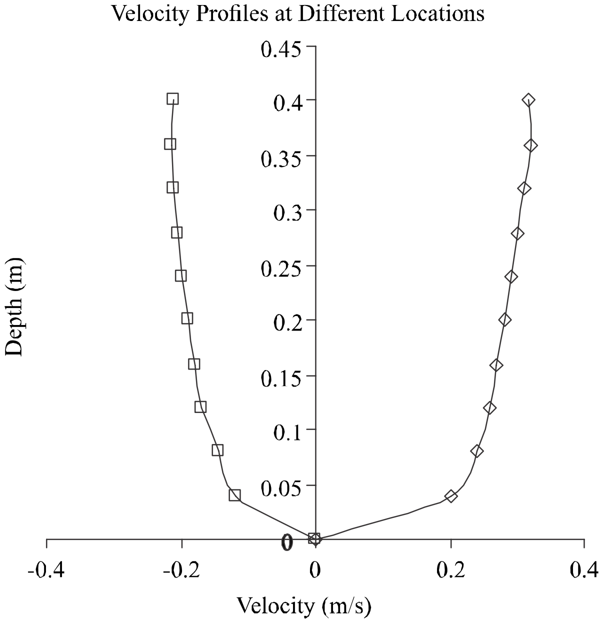
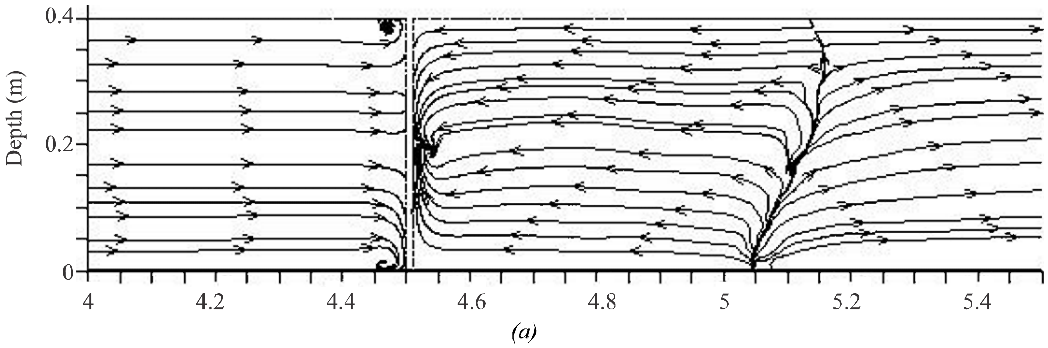


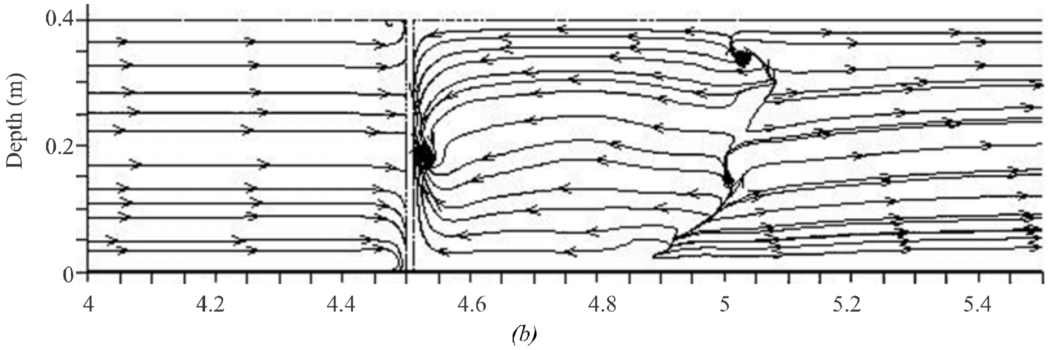
FIG. 5. VELOCITY PROFILES UPSTREAM AND DOWNSTREAM OF THE DIKE

3.4 Turbulent Kinetic Energy

The TKE (Turbulent Kinetic Energy) is a measure of turbulence in the flow. The following diagram (Fig. 8) is representing the distribution of TKE over different sections both upstream and downstream the spur-dike. The intensity of turbulence was very low when the flow was undisturbed (Fig. 8(a-b)), however the moment it



(a)



(b)

FIG. 6(a-b). STREAMLINES ON THE LONGITUDINAL VERTICAL PLANES THROUGH THE DIKE crossed the dike the turbulence increased tremendously as is clear from Fig. 8 (c-e). The turbulence is maximum in regions just behind the dike as compared to the rest part of the cross section.

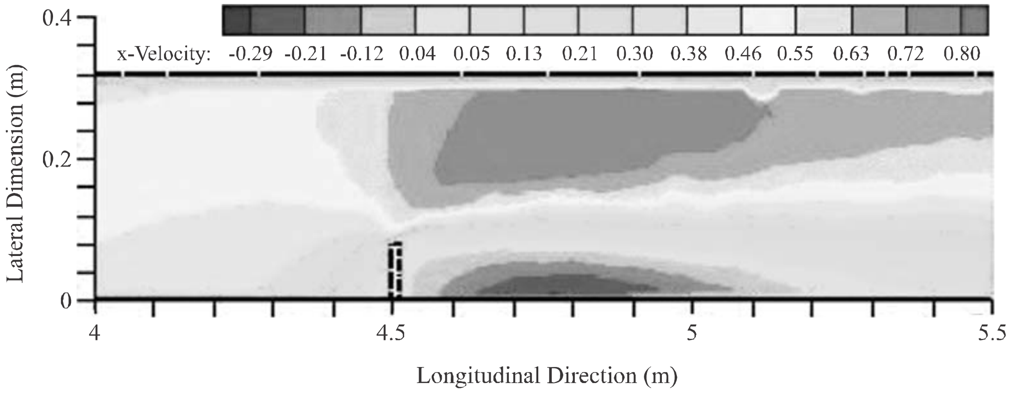
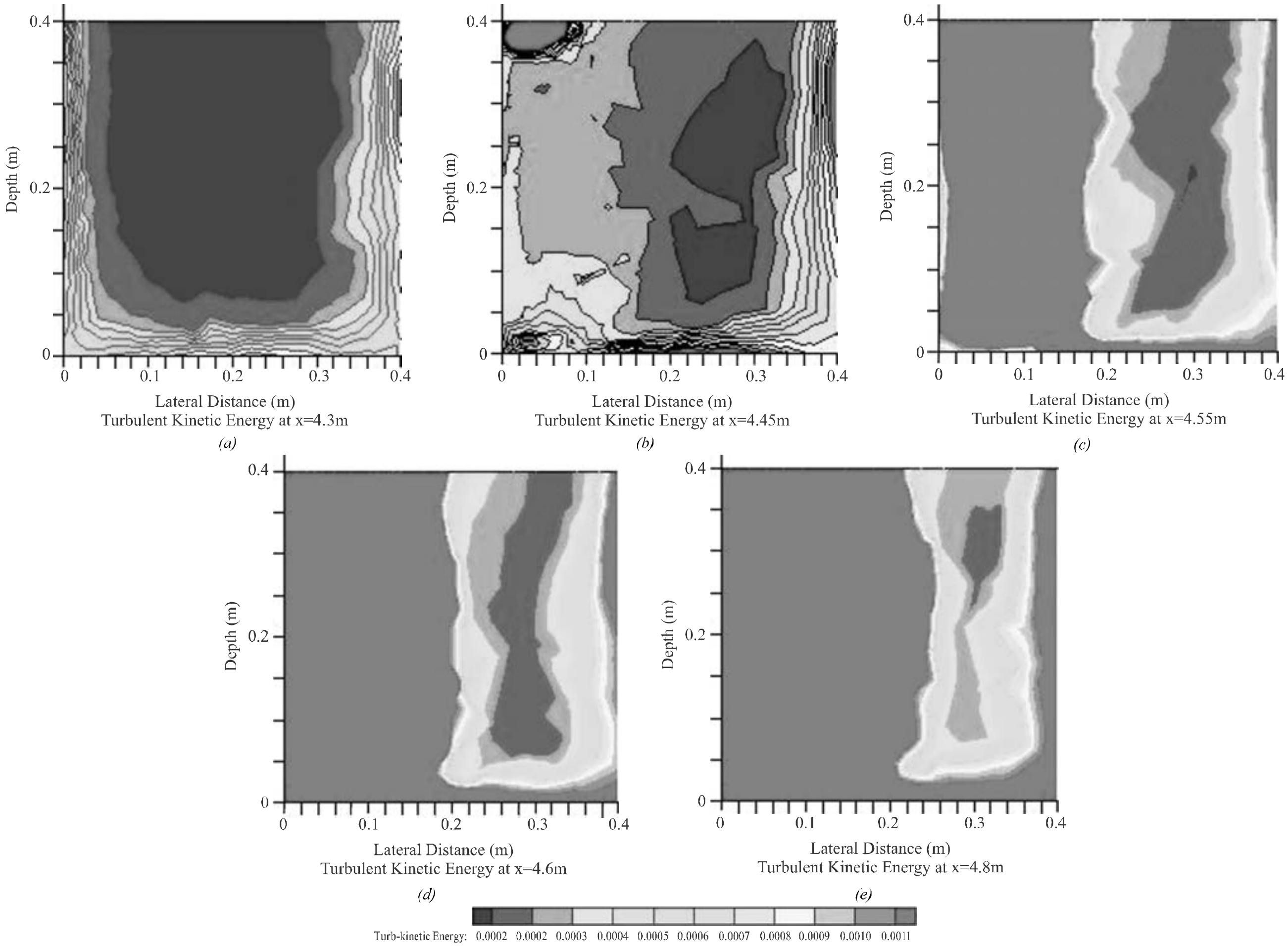


FIG. 7. VELOCITY CONTOURS OVER A HORIZONTAL LONGITUDINAL PLANE AT Y=0.27m FROM BED



(a) *(b)* *(c)* *(d)* *(e)*

FIG. 8(a-e). TURBULENT KINETIC ENERGY OVER DIFFERENT SECTIONS ALONG THE CHANNEL

4. CONCLUSION

The paper was aimed at enhancing the understanding of flow characteristics in the presence of a spur dike in an open channel. It was revealed that primary velocities exhibit flow separation and recirculation just behind the spur dike which might lead to severe erosion of bed in those regions. It was explored through simulation that the reversal of flow happens and is directed downward near the surface and close to the bed. Similarly investigation indicated that TKE is also affected considerably due to presence of spur-dike. It was maximum behind the spur-dike and distributed over the major part of the cross-section for some distance (up to around three times the depth of flow) behind the dike.

ACKNOWLEDGEMENTS

This numerical simulation was performed by using the modeling facilities available at Department of Civil Engineering, UET (University of Engineering & Technology), Taxila, Pakistan. The authors are highly acknowledged to Higher Education Commission, Pakistan for providing these facilities at UET, Taxila, and Lahore, Pakistan.

REFERENCES

[1] French, R.H., "Open Channel Hydraulics", McGraw-Hill Publishing Company, Singapor, 1985.

[2] Yossef, M.F.M., ''Morphodynamics of Rivers with Groynes'', Delft University Press, Delft, Chapter-2, Netherland, 2005.

[3] Tu, J., Yeoh, G.H., and Liu, C., "Computational Fluid Dynamics, A Practical Approach", Elsevier Publishing Company, New York, USA, 2008.

[4] Jennifer, G.D., "Mean Flow and Turbulence Around a Laboratory Spur Dike'', ASCE Journal of Hydraulic Engineering, Volume 135, No. 10. pp. 803-811, USA, October, 2009.

[5] Ahmad, F., and Rajaratnam, N., "Observation of Flow Around Bridge Abutment", ASCE Journal of Engineering Mechanics, Volume 126, No. 1. pp. 51-59, USA, January, 2000.

[6] Chrisohoides, A., Sotiropoulos, F., Sturm, T.W., Yarnell, D.L., and Nagler, F.A., "Coherent Structures in Flat Bed Abutment Flow: Computational Fluid Dynamics Simulations and Experiments", ASCE Journal of Hydraulic Engineering, Volume 129, No. 3. pp. 177-186, USA, September, 2003.

[7] Koken. M., and Constantinescu, G., "An Investigation of Flow and Scour Mechanism Around Isolated Spur-Dikes in a Shallow Open Channel Flows”, Journal of Water Resources Research, Volume 44, No. 8, pp. W08406, USA, September, 2008.

[8] Kwan, R.T.F., and Melville, B.W., "Local Scour and Flow Measurement at Bridge Abutments", IAHR Journal of Hydraulic Research, Volume 32, No. 5, pp. 661-674, Netherland, October, 1994.

[9] Barbhuiya, A.K., and Dey, S., "Vortex Flow Field in a Scour Hole Around Abutments", International Journal of Sediment Research, Volume 18, No. 4, pp. 310-325, China, October, 2003.

[10] Mayerle, R., Toro, F.M., and Wang, S.S.Y., "Verification of Three Dimensional Numerical Model Simulation of Flow in the Vicinity of a Spur-Dike", IAHR Journal of Hydraulic Research, Volume 33, No. 2, pp. 243-256, Netherland, April, 1995.

[11] Tang, X.L., "Experimental and Numerical Investigations of Secondary Flows and Sedimentations Behind a Spur-Dike'', Journal of Hydrodynamics, Volume 19, No. 1, pp. 23-29, January, 2009.

[12] Kimura, I., and Hosoda, T., "A Non-Linear k-ε Model with Realizeability for Prediction of Flow Around Bluff Bodies", International Journal for Numerical Methods in Fluids, Volume 43, pp. 818-837, June, 2003.

[13] Ho, J., Yeo, H.K., and Ahn, W.S., "Numerical Modeling Studies for Flow Pattern Changes Induced by Single Gryone'', 32nd International IAHR Congress, Volume 32, pp. 1234-1242, Italy, July, 2007.

[14] User Guide Fluent 13. Lebnon: FLUENT Incorporated Lebanon, New Hamshire, USA, 2011.

[15] Zhang, H., Nakagawa, H., Kawaike, K., and Baba, Y., "Experiment and Simulation of Turbulent Flow in Local Scour Around a Spur-Dike'', International Journal of Sediment Research, Volume 24, No. 1, pp. 33-45, China, January, 2009.