Modelling of flow pattern in an open channel with sidewall obstruction

S Mulahasan 1*, S Al-Osmy 1, and S Alhashimi 1
1 Water Resources Engineering Department, Mustansiriyah University, Baghdad-Iraq.
*E-mail: irrigation_2010@yahoo.com

Abstract. To predict flow pattern of shallow obstructed open channel flows, laboratory experiments of flow around a vertical emergent sidewall abutment (lateral constriction) in an open channel flow is simulated numerically and results of water surface and vertical profile of velocity are compared with the numerical results. The turbulence kinetic energy at the separation zone and bed shear stresses are also investigated. A computational fluid dynamic (CFD) numerical tool and volume of fluid (VOF) were used to predict water surface profiles. Results showed a very good agreement between the numerical and the experimental results of the water surface profiles. The numerical findings highlighted an increase in the velocity around the structure of two times the average flow velocity. The maximum pressure was observed in front of the constriction increases with increasing discharges. The vertical velocity profiles upstream the sidewall abutment at the three selected locations showed reasonable results. The maximum turbulence kinetic energy is shown at the zone of separation when the flow passes around the nose with high velocity. In this research, numerically it was found that the estimated bed shear stresses are 2-3 times the mean bed-shear stress of incoming flow. While, previous researchers finding showed a maximum value of bed shear stress near the leading edge 3.63 to 5 times the bed shear stress of the incoming flow. Because of the wide range variation in calculating bed shear stresses by the previous researchers, so our research aims to calculate it using CFD and compare it with others. Also, dynamics of the flow around the obstacle was considered in this research under uniform flow conditions for an aspect ratio ranged 5 to 10.

Keywords: Sidewall abutment; Computational fluid dynamics (CFD); Shear stresses. Coherent structures.

1. Introduction
A sidewall obstruction is a hydraulic structure extended from the river bank or a stream. The sidewall hydraulic structure deflects the flow in a narrower way causes an increase in the velocity [1], change the flow pattern locally [2], back water [3-5] and flow circulation of vortices downstream [6-9]. The flow around a sidewall is characterized as a 3-dimensional, very complex unsteady turbulent flow [10-15]. The flow downstream the abutment highlighted the circulation of eddies in the wake zone [14, 15]. Flow characteristics of an obstructed open channel by a hydraulic structure have been studied extensively experimentally and numerically through a number of research papers [16-19]. The flow around a vertical spur dike or around bridge abutments having many similarities should be like [20, 21].

Ettema and Muste (2004) investigated the influence of length scales of flow field around small-scale hydraulic models placed in a flatbed channel [22]. Many physical experimental models were
investigated for different lateral constriction geometries [3, 10, 20, 23, 24]. Numerical approaches were used to simulate flow around sidewall obstacle through a number of research papers [2, 9, 12, 25-29].

Flow fields around sidewall vertical-obstruction within scour holes have been investigated by a number of researchers [11, 14, 15, 30]. Counter clockwise vortices at the score hole were observed at the upstream side of the lateral due to pressure gradients [30-32]. Turbulent intensity around a spur dike has been studied by Duan and Xiufang, et al [8, 29]. Velocity measurements at the inner side of an abutment leading edge in an open channel flow are related to the blockage ratio and friction factor. The results highlighted that for a contraction ratio of 0.3, the velocities reach 1.75 times the incoming velocities [23].

Many topics with a sidewall obstruction have been investigated, for example flow resistance by a single obstacle [25, 33]; the influence of the flow field on pollutant transport [34], enhancing aquatic habitat in the Middle Mississippi River [35]. Horseshoe vortex was numerically investigated [12]. The dynamics of the vortex around a scoured bridge abutment was experimentally [1, 32] and numerically [2, 11] investigated. Mainly, in a straight channel obstructed by a sidewall abutment it was found numerically that coherent structures have processed in the in the scour zone [10, 31].

Effects of the sidewall obstacle lengths on turbulent flow structures are investigated [36]. It was found that in the flow direction, the main vortices for short obstacle disappear over a much shorter distance than those of long and medium structure. Characteristics of forced pools is influenced by the shape of the obstruction [37]. Experimentally, in turbulent flow it was found that flow pattern downstream the obstacle affected by the Froude number [38]. An increase in Froude number strengthen the flow jet.

Bed shear stress around the sidewall abutment was experimentally investigated by a number of researchers [10, 24, 32, 39]. Bed shear stresses at the upstream leading edge of the sidewall abutment was studied by Rajaratnam & Nwachukwu and found to be five times the bed shear stresses in the incoming flow zone [40]. Shear stresses through a sidewall abutment with three blockage ratios (0.1, 0.2, and 0.3) were investigated [24]. Bed shear stresses at these abutments was found 10 times the bed shear stress of incoming flow zone. Also, it was found that as the contraction is increased the flow velocity is increased 1.5 times the flow incoming velocity. Another study of the bed shear stresses near the leading edge of the obstacle which observed an increase of 3.63 times the upstream bed shear stress [10]. Bed shear stresses values around the leading edge of the lateral structure were observed, while in the mixing zone bed shear stresses is increased with the length of the structure. Azinfar, and Kells investigated bed shear stresses in the vicinity of a spur dike [25]. It was found that an increase in bed shear stress values is due to the friction. Estimating the bed shear stresses from the Reynolds shear stresses was investigated [8, 23]. The observed bed shear stress was estimated to be three times larger than the mean shear stress of approaching flow. Jeon and Kang investigated bed shear stresses in bridge models [41]. It was found that the highest values of bed shear stress are close to nose of the abutment and downstream of the abutment.

The aim of this study is numerically to characterize the turbulent flow close to nose of the physical model sidewall structure and in front of the structure in an open channel flow. In addition, dynamics of the free surface profiles was simulated and compared to the experimental data. Bed shear stresses were calculated by the CFD and compared to previous studies.

2. Methods, materials and flow conditions

2.1 The physical sidewall model

Laboratory experiments were achieved in a flume with length of 10 m, and 30 cm its width and depth at Cardiff University. The flume was set to a bed slope of 0.001. Under a uniform flow condition, the sidewall abutment was inserting at a one side of the flume at section were a fully developed flow conditions were achieved. The model has a square cross section of 9 cm side length and 10 cm height (see Fig.1). Three flow rates were examined (2, 3 and 4 litre/sec) with corresponding uniform flow depths of (3 cm, 4.5 cm and 5.9 cm) respectively. An aspect ratio $A_r = W/H$, where $W$ is the flume width
and H is uniform flow depth was chosen within a range of 5 to 10. The flow conditions are explained in Table 1. Calculation of the Reynolds numbers and Froude numbers are based on the flow characteristics such as the bulk velocity, \( U_b \), and the uniform flow depth \( H_{uni} \), so \( Re = \frac{U_b H_{uni}}{\nu} \), and \( Fr = \frac{U_b}{(gH_{uni})^{0.5}} \) respectively. It was seen that from Table 1 that the flow is turbulent and subcritical. The blockage ratio is defined as the length of the lateral constriction (L = 9 cm) perpendicular to flow direction divided by the width (W = 30 cm) of the open channel. The structure was located at 2.1 m from the channel inlet.

![Figure 1. Schematic Plan View of Non-submerged Lateral Constriction Showing the Main Zones.](image)

**Table 1. Flow Conditions of the Experimental Work.**

| Discharge Q (l/s) | Water depth H (cm) | Bulk velocity \( U_b \) (m/s) | Reynolds number \( Re = \frac{U_b H_{uni}}{\nu} \) | Froude number \( Fr = \frac{U_b}{(gH_{uni})^{0.5}} \) |
|-------------------|-------------------|-----------------|------------------|------------------|
| 2.0               | 3.0               | 0.222           | 6560             | 0.409            |
| 3.0               | 4.5               | 0.222           | 9930             | 0.334            |
| 4.0               | 5.9               | 0.225           | 10959            | 0.295            |

Where, \( Q = \) discharge, \( H_{uni} = \) uniform flow depth, \( U_b = \) bulk velocity \( (= Q/A) \), where \( A \) is cross sectional area, \( Re = \) Reynolds number, \( \nu = \) Kinematics viscosity, \( Fr = \) Froude number, and \( g = \) gravitational acceleration.

Along the mid distance between the flume sidewall and the inner edge of the sidewall abutment the water depths were measured at intervals of 0.1 m in the sections \( 1.1 \text{ m} \leq x \leq 2.05 \text{ m}, 2.29 \text{ m} \leq x \leq 3.19 \text{ m} \) and at interval of 0.01 m in the section \( 2.05 \text{ m} \leq x \leq 2.29 \text{ m} \), while it was measured at interval of 0.05 m in the sections \( 2.05 \text{ m} \leq x \leq 2.10 \text{ m} \) and \( 2.24 \text{ m} \leq x \leq 2.29 \text{ m} \). A point gauge with an accuracy of \( \pm 0.1 \text{ mm} \) was used in measuring the water depths.

2.2 Simulate the model using CFD.

Application of CFD code to flows in open channels depends on employing the methodology of turbulent flow model with two phase free surface flow. Averaged Navier–Stokes equations with turbulence models are employed. The water level in each cell is predicted using volume of fluid (VOF) model, which is used in simulating the problems in hydraulic engineering problems with free surface interface between air and water [42].

The governing equations for 3D unsteady incompressible flows around a lateral constriction are the continuity and Navier-Stoke equations. These equations are used to simulate 3D unsteady incompressible flows around a lateral constriction using ANSYS FLUENT software as follows [43]:

\[
\frac{\partial p}{\partial t} + \sum_{i} \left( \rho u_i \frac{\partial u_i}{\partial x_i} \right) = 0 \tag{1}
\]

and,

\[
\frac{\partial}{\partial t} (\rho u_i) + \sum_{i} \left( \rho u_i u_j \frac{\partial u_i}{\partial x_i} \right) = -\sum_{i} \left( \frac{\partial p}{\partial x_i} \frac{\partial u_i}{\partial x_i} \right) + \mu \left( \sum_{i} \left( \frac{\partial u_i}{\partial x_i} \frac{\partial u_j}{\partial x_i} \right) \right) + \rho g_i + \mathbb{F} \tag{2}
\]
2.2.1 Turbulence flow models

Fluctuating velocities in turbulent flows can be transported in terms of momentum kinetic energy and its concentrations which cause mixing due to its fluctuating. Due to its small scale with high frequency cost of direct simulation is expensive. So, to solve this problem, time-averaged ensemble method or any other method can be used to remove the small scales fluctuations to obtain a modified equation [44]. Turbulence models such as \((k – \varepsilon)\) and \((k – \omega)\), Reynolds Stress Model (RSM) were applied as a stable converging solution and their results can be compared with the experimental data of flow around sidewall abutment.

2.2.2 \(K – \varepsilon\) Turbulence Model

In this model, the flow is assumed fully turbulent and the effects of viscosity are negligible. The two equations used to solve such problems are turbulent kinetic energy equation \((k)\) and the dissipation rate of turbulent energy equation \((\varepsilon)\) and written as follows, [45];

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left(\mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j}\right) + G_k - \rho \varepsilon \tag{3}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j}\right) + G_{1\varepsilon} \frac{\varepsilon}{k} G_k - G_{2\varepsilon} \frac{\varepsilon^2}{k} \tag{4}
\]

Where \(\mu_t\) is the eddy viscosity which can be computed by equation (5);

\[
\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon} \tag{5}
\]

Where, \(G_i\) is the production of turbulent kinetic energy. The constants \(C_{\mu} = 0.09\), \(G_{1\varepsilon} = 1.44\) and \(G_{2\varepsilon} = 1.92\). While, the Prandtl turbulent numbers for the turbulence model \(\sigma_k\) and \(\sigma_\varepsilon\) equal to 1.0, and 1.3, respectively.

2.3 Boundary conditions

The issue closely associated with constructing the geometry is setting up the grid, which is the pre-processor available in the ANSYS FLUENT code version 15. As the gridding size has direct effect on the time consuming of computation for CFD modelling beside accuracy and agreement degree with the results of the experimental work. The obtaining initial results needs to examine a number of grids with various sizes and comparing to experimental data. Implementing the meshing sensitivity analysis, a mesh was considered with a total of 576664 nodes and 548255 cells. To regulate solution field, grid selection was based in such a manner that in line with the flow direction. Figure 2 shows the mesh of solution field in our simulation to case studies. Then FLUENT software was applied for CFD set up solution. Finite volume method (FVM) was considered in solving 3-D flow equations. It takes into accounts the discrete control volumes to show the entire flow field. To reach acceptable convergence point, the user employed algorithms in solving algebraic discrete equations. In our study, the models for different discharge have the same boundary conditions; the flow domain was selected as symmetry at the top while the bottom and the two sides as a no-slip wall, the upstream and downstream conditions were substituted in pressure, its values were corresponding to those experimental measurements. The upstream condition is the water depth in front of the lateral constriction, while the downstream of the channel is the pressure outlet condition (see table 2). The pressure-velocity coupling algorithm (PISO) and the PRESTO discretization scheme were applied for transient and pressure simulations respectively. Modified High Resolution Interface Capturing (HRIC) method was applied for the volume fraction, while QUICK and second-order upwind was used for momentum and turbulent kinetic energy discretization schemes respectively [46]. The 3D flow was simulated with VOF to model the free surface two-phase flow and solving continues until the residual reached \(10^{-6}\).  

\[ 

\]
3. Results and discussion

In this study, flow field around a sidewall structure (lateral constriction) was investigated. A laboratory flume of rectangular cross section (0.3 m X 0.3m) wide and deep was used and the lateral constriction was mimicked by a plastic cube of 9.0 cm side. Three flow rates (2, 3 and 4 liter/sec) with corresponding uniform flow depths of (3 cm, 4.5 cm and 5.9 cm) were examined. A numerical approach ANSYS FLUENT was used to simulate the flow field around a lateral constriction. Velocity contours, streamlines, pressure distribution, total kinetic energy (TKE), vertical velocity profiles and bed-shear stresses were presented. Predicting water surface profiles was achieved using volume of fluid method. Figure (3) showed the findings of the water surface profiles using a numerical approach (CFD) in comparison to the experimental water surface measurements. It can be seen a very good agreement was achieved by the numerical simulation to the experimental data.

Figure (4) provided spatial flow velocity distribution contours in the stream-wise direction in the vicinity of the obstacle for three different discharges. The maximum velocity is shown at the zone of separation when the flow passes around the nose. The lowest values of velocities are observed in front and behind the sidewall structure. This Figure highlights an increase in the velocity around the structure two times the averaged flow velocity $U_b$.

Streamlines have been drawn in Figure (5). It shows that the contracted streamlines are similar in trend for all values of the examined discharges. Streamlines around the structure provided a state of the flow velocity which increases when the flow passes from the nose of the structure, while the velocity behind the sidewall abutment is decreased. There is a separation zone of flow generated behind the constriction, but reattachment starts when the main flow confluences with the flow behind the obstacle.
which is called point of reattachment as indicated by Hoffmans & Pilarczyk [47]. Due to the existence of high-pressure zone found in front of the lateral constriction, the flow accelerates around the leading edge. Clockwise eddies behind the lateral constriction located within the wake zone were monitored.

Figure 3. Numerical and Experimental Results of Water Surface Profiles for Various Discharges (a) $Q = 2$ liter/sec, (b) $Q = 3.0$ liter/sec and (c) $Q = 4$ liter/sec.
Figure 4. Velocity Contours Highlight an Increase of Flow Velocity around the Lateral Constriction for Various Discharges (a) $Q = 2$ liter/sec, (b) $Q = 3.0$ liter/sec and (c) $Q = 4$ liter/sec.

Variation the total pressure of flow domain is shown in Figure 6. The maximum pressure was observed in front of the lateral constriction which increase with increasing discharges. Due to the water level differences between the upstream and downstream sides of the lateral constriction, a pressure difference is created between the two sides with high pressure at the upstream side. The results constitute with findings of Mioduszewski et al. [48].
Figure 5. Streamlines around a Lateral Constriction in CFD Numerical Modeling Showing Flow Zones around the Structure for Different Discharges (a) $Q = 2.0$ liter/sec, (b) $Q = 3.0$ liter/sec and (c) $Q = 4.0$ liter/sec.

Figure (7) shows the turbulence kinetic energy for the examined discharges. The maximum turbulence kinetic energy is shown at the zone of separation when the flow passes around the nose with high velocity. The zone of separation is increasing with increasing discharge. Generally, downstream the constriction the flow has low turbulence. While, near the bed of the constriction, maximum turbulent kinetic energy was observed.
Figure 6. Distribution of Pressure around Lateral Constriction in CFD Numerical Modeling for Various Discharges (a) $Q = 2.0$ liter/sec, (b) $Q = 3.0$ liter/sec and (c) $Q = 4.0$ liter/sec.
Distribution of bed shear stress around the lateral constriction was investigated. Generally, maximum bed shear stress was observed in the main flow zone near to the nose of the constriction, which where the flow is blockaded by the lateral constriction. Fig. 8 shows the effect of discharge on bed shear stress. It was observed that higher velocity and bed shear stresses around the lateral constriction is with smaller discharge. The maximum shear stress is observed to reach 1.455, 1.325, and 1.276 Pa for the discharges of 2, 3, and 4 litre/sec respectively. Estimated bed shear stresses values from this study were compared to the researcher's findings. For example, Duan (2009) estimated maximum bed shear stresses of 2-3 times the mean bed shear stress of approaching flow [8]. While, Rajaratnam & Nwachukwu showed that the bed shear stress near the leading edge of the lateral constriction has a maximum value of five times the bed shear stresses in the incoming flow zone [40]. However, Ahmed & Rajaratnam showed that the bed shear stress near the leading edge of the structure is increased by 3.63 times the upstream bed shear stress [10]. It was shown that, near the tip of the lateral constriction, velocity and shear stress are increased, while in the zone of recirculation which locates behind the lateral constriction, bed shear stress decreases.
To understand the flow behaviour in front of the sidewall structure, three arbitrary points located at 10 cm upstream the obstacle were selected. One of these points is located in front the abutment, the second point along the junction edge between the sidewall structure and the free flow and the third point is upstream the structure in the free stream (see Fig. 1). The vertical flow velocity and its corresponding the total kinetic energy TKE and the normalized TKE of the flow were modelled as shown in Figures 9, 10 and 11 respectively. The evaluation included all the tested discharges. Vertical velocity profiles at different locations as indicated in red colour 1, 2 and 3 in front of the lateral constriction are selected as shown in Fig (1). In Fig. 9 shows the results of the velocity profile in the main channel (point 3) indicated the highest value (as expected) and it decreased towards point 2 and 1. Point 1 is completely in front the bluff body (lateral constriction) reveals the lowest velocity in comparison to point 2 and 3. Maximum velocities were monitored in the range of (0.17-0.23) m/sec, (0.19-0.265) m/sec, and (0.213-0.265) m/sec at point 1 to 3 for the discharges 2, 3 and 4 litre/sec respectively. This leads to sediment concentration at the leading-edge area due to the small values of velocity in front the bluff at body point 1.

Figure 8. Bed Shear Stress Distribution for Various Discharges (a) Q = 2.0 litre/sec, (b) Q = 3.0 litre/sec and (c) Q = 4.0 litre/sec.
Figure 9. Vertical Velocity Profiles at Points 1, 2 and 3 in front of the Structure for Various Discharges (a) $Q = 2.0$ litre/sec, (b) $Q = 3.0$ litre/sec and (c) $Q = 4.0$ litre/sec.

Figure (10) provides the numerical solution of the total kinetics energy TKE at different locations in front the sidewall abutment corresponding to the vertical velocity profiles at the same locations. It appeals that total kinetic energy at point 1 refers to the eddy formation in front the obstacle which responsible to sediment accumulation. Highest TKE values are due to highest discharges. Figure 10 appeals that the maximum peak values for the different discharges were in the range of $(0.00315-0.0039)$, $(0.0033-0.005)$, and $(0.0036-0.0052)$ m$^2$/s$^2$ at points 3, 2 and 1 and the measured values were
at a depth of 0.08 m, 0.09 m and 0.13 m from the bed of the channel respectively. Finally, the hydraulic analysis of flow was extracted in the form of normalized total kinetic energy with the bulk velocity, $U_b$ as shown in Figure (11).

\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure10.png}
\caption{Total Kinetic Energy of Flow at Points 1, 2 and 3 in front of the Structure for Various Discharges (a) $Q = 2.0$ litre/sec, (b) $Q = 3.0$ litre/sec and (c) $Q = 4.0$ litre/sec.}
\end{figure}
Figure 11. Normalized Total Kinetic Energy of Flow at Points 1, 2 and 3 in front of the Structure for Various Discharges (a) $Q = 2.0$ litre/sec, (b) $Q = 3.0$ litre/sec and (c) $Q = 4.0$ litre/sec.
4. Conclusions
Flow field around a vertical non-submerged sidewall abutment in an open channel was simulated numerically and results of water surface and vertical profile of velocity are compared with the numerical data. The results of the water surface profiles manifest very good agreements between the numerical and the experimental data. The simulation showed that an increase in the velocity around the structure of two times the averaged flow velocity, $U_b$, and the lowest values of velocities were observed at upstream and downstream the sidewall abutment. High pressure zone was shown in front of the lateral constriction, the flow accelerates around the leading edge. The maximum pressure was observed in front of the constriction which increases with increasing discharges. The maximum turbulence kinetic energy is shown at the zone of separation when the flow passes around the nose with high velocity. The maximum bed shear stress is observed to reach (1.455, 1.325, and 1.276 Pa) for the examined discharges (2, 3, and 4 litre/sec) respectively. Estimated bed shear stresses values from this study were compared to the researcher’s findings. For example, Duan (2009) estimated maximum bed shear stresses were about 2-3 times the mean bed shear stress of incoming flow. Rajaratnam and Nwachukwu (1983) showed that the bed shear stress near the leading edge of the structure shows a maximum value reaches five times greater than that the bed shear stress of the approaching flow. However, Ahmed and Rajaratnam (2000) estimated bed shear stress near the leading edge of 3.63 times bed shear stress of approaching flow. Clockwise eddies behind the lateral constriction located within the wake zone were observed. Dynamics of the upstream wake zone was examined by vertical velocity profiles and kinetic energy of some arbitrary selected points. Minimum velocities were monitored in front zone area which responsible to sediment accumulation. These results will help engineers and scientists in the field of river training structures to better understand.

Acknowledgments
The authors wish to thank all staff of Hydraulic and Hydrology laboratory in School of Engineering at Cardiff University for assistance in performing this research. Many thanks Mustansiriyah University (www.uomustansiriyah.edu.iq) for its encouraging us to perform the research.

5. References
[1] Barbhuiya A.K. and Dey S., 2003 Vortex flow field in a scour hole around abutments, *International Journal of Sediment Research*, Vol. 18, No. 4, pp. 310-325.
[2] Kara S., Stoesser T., Sturm T. W., and Mulahasan S., 2015 Flow dynamics through a submerged bridge opening with overtopping, *Journal of Hydraulic Research*, Vol. 53, No. 2, pp. 186–195.
[3] Launder B. and Spalding B. D., 1974 The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering*, No. 3.
[4] Seckin G. and Atabay S., 2005 Experimental backwater analysis around bridge waterways, *Can. J. Civ. Eng.*, 32: 1015–1029.
[5] Al-Hashimi S. A. M., Al-Osmy S. A. and Mulahasan, S., 2020 Water surface profile and flow pattern simulation over bridge deck slab, *Journal of Engineering Science and Technology*, Vol. 15, Issue 1, pp. 290-304.
[6] Afsalimehr H., Bakhshi S., Ghalichand J., and Sui J., 2014 Effect of vegetated-banks on local scour around a wing-wall abutment with circular edges, *J. Hydrodyn.*, 26(3), 447–457.
[7] Dey S. and Barbhuiya A.K., 2005 Flow field at a vertical-wall Abutment, *Journal of Hydraulic Engineering*, Vol. 131, Issue 12.
[8] Duan J. G., 2009 Mean Flow and Turbulence around a Laboratory Spur Dike, *Journal of Hydraulic Engineering*, Vol. 135, Issue 10.
[9] Yazdi J., Sarkardeh H., Azamathulla H. and Ghani A. 2010 3D simulation of flow around a single spur dike with free-surface flow”, *Intl. J. River Basin Management*, Vol. 8, No. 1, pp. 55–62.
[10] Ahmed F. and Rajaratnam N., 2000 Observations on flow around bridge abutment, *J. Eng. Mech.*, 10,1061/ (ASCE) pp.51–59.
[11] Bressan F., Ballio F. and Armenio V., 2011 LES of turbulence around a scoured bridge abutment, Journal of turbulence, Part of the ERCOFTAC Series book series (ERCO, volume 15), pp 251-256.
[12] Kirkil G. and Constantinescu G., 2010 Flow and turbulence structure around an in-stream rectangular cylinder with scour hole, Water Resources Research, Vol. 46, W11549. doi: 10.1029/2010WR009336.
[13] Koken M. and Constantinescu G. 2006 Investigation of Flow around a bridge abutment in a flatbed channel using large eddy simulation, World Environmental and Water Resources Congress.
[14] Koken M. and Constantinescu G., 2008 An investigation of the flow and scour mechanisms around isolated spur dikes in a shallow open channel, Water Resources Research, Vol. 44, W08406. https://doi: 10.1029/2007WR006489.
[15] Terruzi A., Ballio F., Salon S. and Armenio, V., 2006 Numerical investigation of the turbulent flow around a bridge abutment, River Flow.
[16] Beheshti A. and Ashtiani, B.A., 2010 Experimental Study of Three-Dimensional Flow Field around a Complex Bridge Pier, Journal of Engineering Mechanics, 143.
[17] Molinas A, Kheireldin K and Wu, 1998 Shear stress around vertical wall abutments, Journal of Hydraulic Engineering, Vol. 124, Issue Number: 8, pp: 822-830.
[18] Nagata N., Hosoda T., Nakato T., M. and Muramoto, Y., 2005 Three-dimensional numerical model for flow and bed deformation around river hydraulic structures”, Journal of Hydraulic Engineering.
[19] Tang X., Ding X. and Chen Z., 2006 Large eddy simulations of three-dimensional flows around a spur dike, Tsinghua Science & Technology, 11(1):117-123.
[20] Afzalimeh, H. Moradian M. and Singh V.P., 2018 Flow field around semi-elliptical abutments, Journal of Hydrologic Engineering, 23(2): 04017057.
[21] Koken M., Kirkil G., and Constantinescu G., 2007 Coherent structures in the flow around a bridge abutment and a bridge pier at equilibrium scour conditions. Proc., 32nd Congress of IAHR, Venice, Italy, CORILA, Venezia.
[22] Ettema R. and Muste M., 2004 Scale effects in flume experiments on flow around a spur dike in flatbed channel, J. Hydraul. Eng., 130:635-646, (2004).
[23] Barbhuiya A., and Dey S., 2004 local scour at abutments: a review’s adhan a vol. 29, part 5, pp. 449–476.
[24] Molinas A. and Hafez Y.I., 2000 Finite element surface model for flow around vertical wall abutments, Journal of Fluids and Structures, 14, 711-733, 2000, doi:10.1006/jfls.2000.0295.
[25] Azinfar H. and Kells J. A., 2009 Flow resistance due to a single spur dike in an open channel, Journal of Hydraulic Research, Vol. 47, Issue 6.
[26] Chen L and Jiang JC., 2010 Experiments and numerical simulations on transport of dissolved pollutants around spur dike, Water Science and Engineering, 3(3): 341-353. https://doi:10.3882/j.issn.1674-2370.2010.03.010.
[27] Koken M. and Constantinescu G., 2009 An investigation of the dynamics of coherent structures in a turbulent channel flow with a vertical sidewall obstruction, Physics of Fluids, 21, 085104.
[28] Vaghefi M. Safarpoor Y. and Akbari M., 2017 Numerical Comparison of the Parameters Influencing the Turbulent Flow using a T-shaped Spur Dike in a 90° Bend, Journal of Applied Fluid Mechanics, Vol. 10, No. 1, pp. 231-241.
[29] Xiufanga Z., Pingyi W. and Chengyu Y., 2012 Experimental Study on Flow Turbulence Distribution around a Spur Dike with Different Structure, Procedia Engineering 2012 International Conference on Modern Hydraulic Engineering.
[30] Manes C., and Brocchini M., 2015 Local scour around structures and the phenomenology of turbulence, J. Fluid Mech. vol. 779, pp. 309_324, doi:10.1017/jfm.2015.389 309.
[31] Chrisohoides A., Fotis S F. Terry W. and Sturm T.W., 2003 Coherent Structures in Flat-Bed Abutment Flow: Computational Fluid Dynamics Simulations and Experiments, Journal of Hydraulic Engineering, Volume 129 Issue 3.
[32] Dey, S., 2014 Fluvial hydrodynamics: Hydrodynamic and sediment transport phenomena, Springer, Berlin.
[33] Robertson F. H., and Lane-Serff G. F., 2019 Drag on pairs of square section obstacle in free surface flow”, Physics Review Fluids, 3(12), DOI: 10.1103/PhysRevFluids.3.123802.

[34] Chen L.P and Jiang, J.C., 2010 Experiments and numerical simulations on transport of dissolved pollutants around spur dike, Water Science and Engineering, 3(3): 341-353.

[35] Remo J.W.F., Khanal A., and Pinter, N., 2013 Assessment of chevron dikes for the enhancement of physical-aquatic habitat within the Middle Mississippi River, USA’ Journal of Hydrology 501:146-162. https://doi: 10.1016/j.jhydrol.2013.07.007, (2013).

[36] Koken M. and Constantinescu G., 2011 Flow and turbulence structure around a spur dike in a channel with a large scour hole, Water Resources Research, Vol. 47, W12511.

[37] Thompson D. M. and Carrick C. R. Mc., 2010 A flume experiment on the effect of constriction shape on the formation of forced pools, Hydrol. Earth Syst. Sci. Discuss., 7, 1945–1972.

[38] Liu H. K., Bradley J. N. and Plate E. J., 1957 Backwater effects of piers and abutments. Colorado State University, CER57HKLIO.

[39] Safarzadeh A., Neyshabouri S.A.A., Ghodsian M. and Zarrati A.R. 2010 Experimental study of head shape effects on shear stress distribution around a single groyne, River Flow.

[40] Rajaratnam N. and Nwachukwu B.A., 1983 Flow near groin-like structures, Journal of Hydraulic Engineering, ACSE, 109(3): 463-480.

[41] Jeon J and Kang S 2016 Flume experiments for turbulent flow around a spur dike, J. Korea Water Resour. Assoc. Vol. 49, No. 8, pp. 707-717. https://doi:10.3741/JKWRA.2016.49.8.707.

[42] Hirt C. W. and Nichols B. D. 1981 Volume of fluid (VOF) method for the dynamics of free boundaries, Journal of Computational Physics, Volume 39, No. 1, pp. 201-225.

[43] Choudhury D., 1993 Introduction to the renormalization group method and turbulence modeling, Fluent Inc. Technical Memorandum, TM-107.

[44] Wilcox D. C., 1993 Turbulence Modeling for CFD, DCW Industries Inc., La Canada, California.

[45] Kuhnle R.A., Jia, Y., and Alonso C.V., 2005 Measured and simulated flow near spur dikes, US-China workshop on advanced computational modelling in hydro-science & engineering, September 19-21, Oxford, Mississippi, USA.

[46] ANSYS, ANSYS Fluent User’s Guide, ANSYS, Canonsburg, PA, USA, 2011.

[47] Hoffmans G. J., and Pilarczyk K. W. 1995 Local Scour Downstream of Hydraulic Structures, Journal of Hydraulic Engineering, 121:4. https://doi: 10.1061/(ASCE)0733-9429.

[48] Moduszewski T., Maeno S. and Uema Y., 2003 Influence of the spur dike permeability on flow and scouring during a surge pass, Proc. of International Conference on Estuaries and Coasts, Hangzhou.