Comparison of Different Turbulence Models in Open Channels with Smooth-Rough Bedforms

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

RECEIVED ON 19.03.2013 ACCEPTED ON 05.06.2013

ABSTRACT

The turbulence models play an important role in all types of computational fluid dynamics based numerical modelling. There is no universal turbulence model which can be applied in all the scenarios. Therefore, if a suitable closure model is used in a simulation work, only then the successful numerical modelling will be achieved. This paper presents the evaluation of three turbulence models in numerical modelling of open channel flows having beds comprising of two parallel strips, one being smooth and the other one being rough. The roughness on the rough side of the channel was created with the help of gravels. The turbulence models tested for their suitability in this case were Reynolds stress model, k-ε model and RNG based k-ε model. A structured mesh was used in this simulation work. Grid independence test was also conducted in the simulation. The evaluation of the turbulence models was made through the primary velocity contours and secondary velocity vectors over the cross section of the channel. It was revealed that Reynolds stress model simulated the flow behaviour successfully and results obtained through this model matched very closely to that of the experimental data whereas k-ε model and RNG based k-ε model failed to reproduce the flow field successfully. These results will be helpful for CFD (Computational Fluid Dynamics) modellers in correct selection of the turbulence model in these types of channels.

Key Words: **Bedforms, Roughness, Computational Fluid Dynamics, k-ε Model, Structured Mesh.**

1. INTRODUCTION

A variety of open channels exist in nature. These include straight and meandering with/without floodplains, floodplains with vegetation, channels with lateral variation of bed roughness, converging channels, diverging channels etc. Research has been done and is being done on these types of channels. However, in some cases such as lateral bed roughness variation, less work has been done by some researchers [1-3].

The presence of different types of roughness in lateral direction is a natural process. Most of the natural rivers have this type of roughness. This roughness variation might be in the form of smooth patch and rough boulders, smooth and vegetation patches, boulders and vegetation patch etc. All these are the examples of roughness variation over the cross section of the open channel flows.

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

In the past some of the researchers made experimental [4-6] research work using a number of parallel smooth and rough patches in a laboratory channel. They examined the effect of these patches on different aspects of flow behaviour including primary and secondary velocity fields, turbulent stresses, turbulent kinetic energy and its dissipation rate etc. They concluded that such a scenario is not only a common situation in practical channel but also affects the flow behaviour considerably.

Recently some of the experimental work has been done by few researchers [7] in UK. These researchers used different forms of bed roughness including parallel smooth-roughness bed strips and also checker-board formation of bed roughness. However, most of this work was experimental. The use of numerical modelling in this research topic is very less. Researchers like Vermass [8] used large eddy simulation in his work and studied some of the flow features under such scenarios. The major impact of stripped roughness on flow field results in reduced conveyance capacity of the channel. It means that the discharge calculation made by ignoring these impacts in the formulae will not yield good results and the channel designed with the help of such formulae will result in a poor design. That's why the research over smooth-rough beds is not only important but ignoring it will prove harmful.

In this research work the 3D CFD has been used to evaluate the performance of various turbulence models in open channel flows with heterogeneous bed formations. The numerical code FLUENT [9] has been used. Three turbulence closure models were tested for their suitability in these types of flows. These include Reynolds Stress Model, k-ε model and RNG based k-ε model. The results have been presented in the shape of primary and secondary velocity vectors.

2. EXPERIMENTAL DESCRIPTION

The experimental data used in this research work is that of Michael Jasson [10]. He performed the experiments in the Hydraulics Laboratory of School of Civil Engineering, University of Birmingham, UK. Fig 1 shows the cross-section and plan-view of the experimental set-up. The data was collected by a ADV (Acoustic Doppler Velocities). The installed ADV in the channel has been shown in Fig. 2. The experiments were performed in a 22m long flume. The cross section of the flume was 0.614m. The flow case considered in this work has a discharge value of 40.3 l/sec and corresponding flow depth of 121.6mm. The flow enters into a stilling tank and passes through a baffle designed to eliminate the turbulence effects. It then enters into the channel. A polystyrene board was placed on the water surface immediately after the entrance of the channel to remove the impact of surface waves. The outflow was controlled through a tailgate to get a uniform flow condition.

The smooth side of the channel was prepared by using two layers of smooth plastic sheets each having a thickness of 10mm i.e. a total thickness of 20mm. The rough side was developed by gluing 10mm thick gravel layer on the top of smooth plastic sheets. The 10mm layer of gravel was comprised of gravel having a mean diameter of d_{72}= 10mm. Thus the total thickness of the gravel side is also 20mm as shown in Fig. 1.

3. NUMERICAL MODELLING PROCEDURE

The 3D CFD based numerical code FLUENT has been used in this research work. The mesh generation was done through Gambit [11]. A structured mesh was used in this modelling. The structured mesh is preferable in case of simple straight channels especially when there cross-sections are rectangular. That's why the structured mesh with hexahedral elements has been used in this simulation work. The mesh independence test was conducted by doubling the node numbers in lateral and vertical directions turn by turn and then simultaneously. However, no significant difference was observed in the results due to mesh refinement. The observed difference for the mesh refinement is less than 1% for primary velocity contours. The finally used mesh had the nodes numbers 200x40x8 in the streamwise, lateral and vertical directions. The simulated results were compared with the experimental resulsts of ADV. The 3D continuity and Reynolds averaged Navior-Stokes momentum equations were solved in this numerical modelling. The SIMPLE algorithm had been used for linking pressure with velocity. The default values of under relaxation factors had been used in this work. The solution was supposed to be converged when all the residuals reached at $1x10^{-06}$. The near wall treatment was achieved through standard wall function. The benefit of using this wall function is that turbulence model will not be used in this region and wall function will handle the turbulence in this zone of flow. In this way less computation cost will be required for simulation work and convergence will be achieved in least time. The boundary conditions included velocity value at inlet and pressure at outlet of the channel, symmetry at the free surface, wall boundary condition at the side walls and bed of the channel.

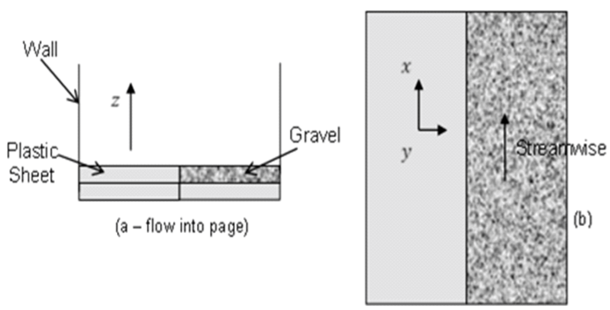

FIG. 1. CROSS-SECTION AND PLAN VIEW OF THE CHANNEL (AFTER MICHAEL JASSON)

FIG. 2. ADV INSTALLATION IN THE CHANNEL (AFTER MICHAEL JASSON)

4. RESULTS AND DISCUSSION

4.1 Reynolds Stress Model

The Fig. 3(a) shows the experimental results for distribution of primary velocity contours over a section of the channel. These are normalized primary velocity contours which have been normalized by the mean inflow velocity i.e. U_m. The right half of the section represents rough patch whereas left half shows the smooth surface. The Fig. 3(b) shows the simulated results for primary velocity contours over the same section by using the Reynolds stress model. The quadratic-pressure strain type of Reynolds stress model has been used in this numerical modelling.

The Fig. 3(a) shows that the velocity values are much smaller over rough side as compared to the smooth side. It has been observed that the simulated results captured all the salient features of the observed experimental results. These include bulging of the velocity values in downward direction at the smooth-rough patch boundaries, upward movement of the primary velocity in the middle of the smooth side i.e. at a distance of around 0.15m from left side of the section. This is an indication of existence of strong secondary circulation at the interaction of smooth-rough beds interaction. The primary velocities are again depressed close to the left bank. This indicates the presence of secondary cells in this region which has also been captured by the secondary flow diagrams as shown in Fig. 4(a-b).

Fig. 4(a-b) shows the distribution of secondary velocity vectors over the same section as given in Fig. 3(a-b). Fig. 4(a) represents the experimental results while Fig. 4(b) depicts numerical results. From these results, it is clear that simulated results matched closely to that of the experimental ones and that numerical model has captured all the salient features of the experimental data. The slow moving water close to the rough bed moves towards the smooth side while fast moving water at the top is directed towards the rough side. This results in formation of secondary cells which causes bulging of primary velocities. Fig. 5 shows the simulated primary velocity contours overlaid be secondary velocity vectors.

4.2 k-ε Model

Fig. 6(a-c) shows the numerical results for primary and secondary velocity vectors over a cross-section of the channel using k-e model. If we compare this diagram with Fig.3(a) and Fig. 4(a), which are the experimental results, then we can conclude easily that the simulated results are different from the experimental ones. This has happened in case of both primary and secondary velocities. Bulging of primary velocity contours and secondary cells over the section have not been captured by the k-ε turbulence model. Similarly the primary velocity values predicted by k-ε turbulence model is slightly higher than Reynolds stress model

The secondary cells have not taken the shape which will show an increase in the bed shear stresses at the junction of the smooth-rough bed as was shown by Reynolds stress model and experimental results of Figs. 3-4. The maximum primary velocity contours occurred totally over the smooth side and did not show the presence of any secondary velocity cells over this smooth side. Again this is against the experimental results.

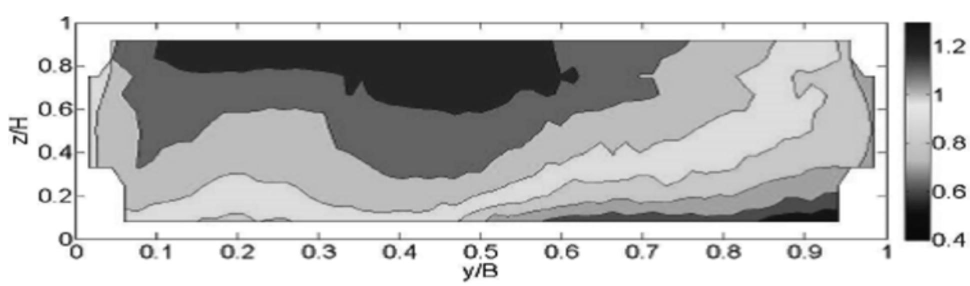

FIG. 3(a). EXPERIMENTAL PRIMARY VELOCITIES

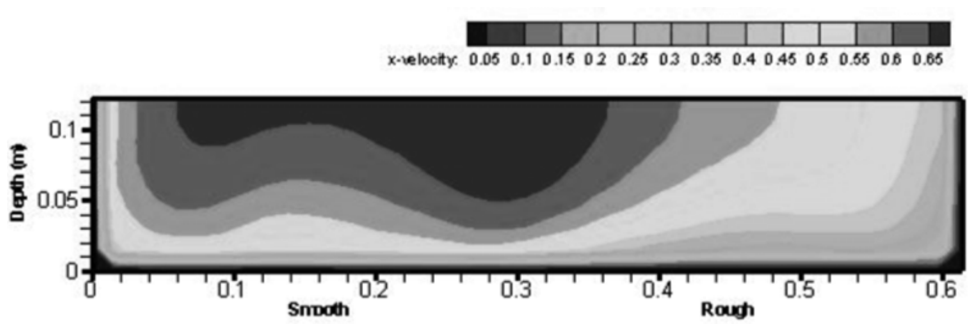

FIG. 3(b). SIMULATED PRIMARY VELOCITIES

4.3 RNG k-ε Model

The Fig. 7 shows results obtained through RNG based k-ε turbulence model. The results produced by the numerical model FLUENT with the help of RNG k-ε model are also poor. They deviated too much from the experimental results of Fig. 3(a) and Fig. 4(a). It also failed to capture secondary cells and bulging of primary velocity contours over smooth-rough bed interaction regions. The maximum primary velocity values fell entirely in a small region of smooth side of bed which ranged from 0.1-0.2m from left side of the channel.

Although the results predicted by k-ε model and RNG k-ε models are very similar but these are entirely different from the experimental ones. The k-ε and RNG k-ε model are based upon the turbulence isotropy which means that turbulent behaviour in an open channel is isotropic i.e. same in all directions, however this is not the situation for straight open channels. The turbulence in these channels is anisotropic i.e. different in different directions due to wall effects and presence of free surface. This might be one of the reasoning for failure of k-ε and RNG k-ε in producing the required results in this straight heterogeneous channel.

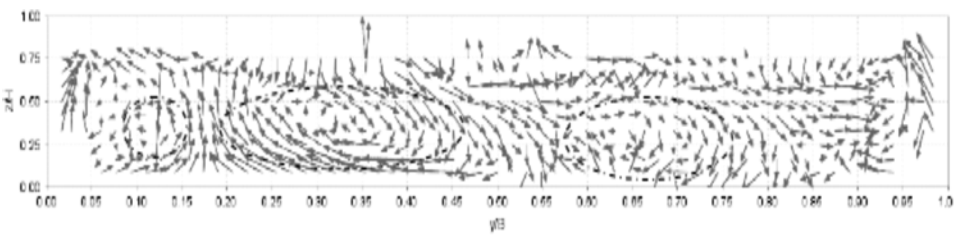

FIG. 4(a). EXPERIMENTAL SECONDARY VELOCITY VECTORS

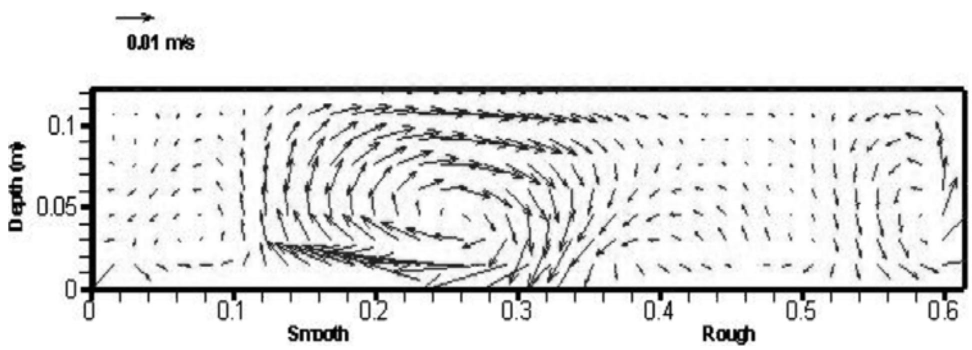

FIG. 4(b). SIMULATED SECONDARY VELOCITY VECTORS

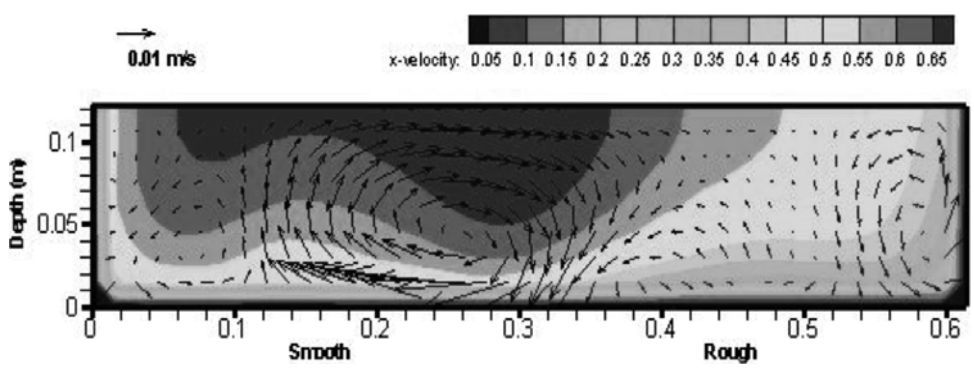

FIG. 5. PRIMARY VELOCITY CONTOURS AND SECONDARY VELOCITY VECTORS

5. CONCLUSION

This research work proved that in case of open channels with lateral bed roughness changes over it cross section, the choice of the turbulence model is very important and if a wrong model is selected then simulated results might be totally wrong. In this case, the k-ε and RNG k-ε model failed to capture the flow features in the channels under investigation but Reynolds stress model simulated very well. So in such a scenario the Reynolds stress model should be adopted rather than k-ε or RNG k-ε turbulence model. It is expected that this research work will help river flow modellers in selection of suitable turbulence model for flow prediction and design purposes in situations which resembles this work.

ACKNOWLEDGEMENTS

The first author is highly acknowledged to Higher Education Commission, Pakistan, for providing computational modelling facilities at University of Engineering & Technology, Taxila, Pakistan, which were exploited in this research work.

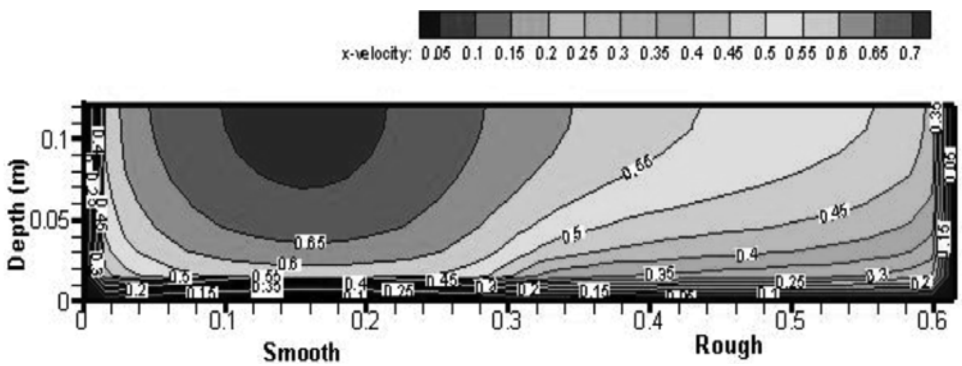

FIG. 6(a). PRIMARY VELOCITY CONTROUS

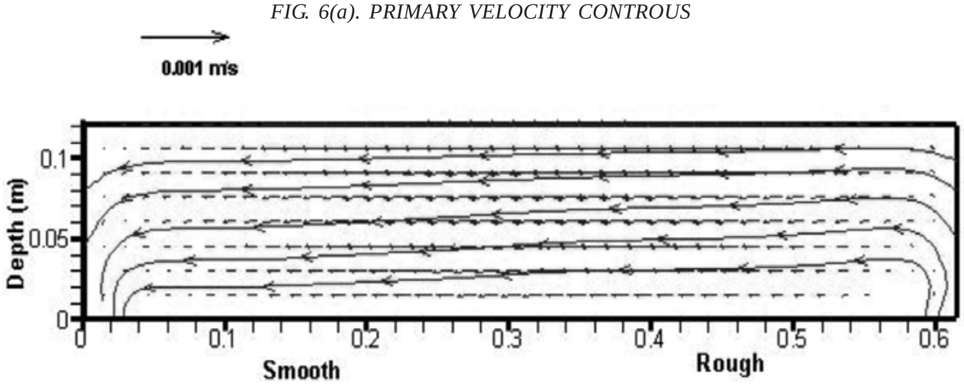

FIG. 6(b). SECONDARY VELOCITY VECTORS

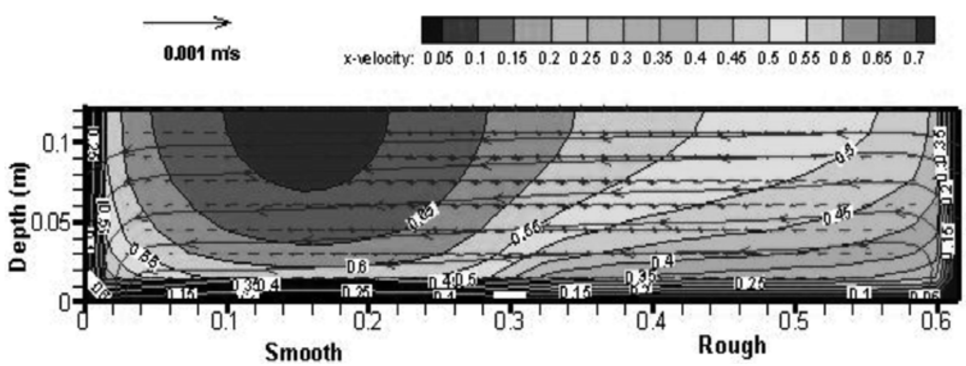

FIG. 6(c). PRIMARY VELOCITY CONTOURS AND SECONDARY VELOCITY VECTORS

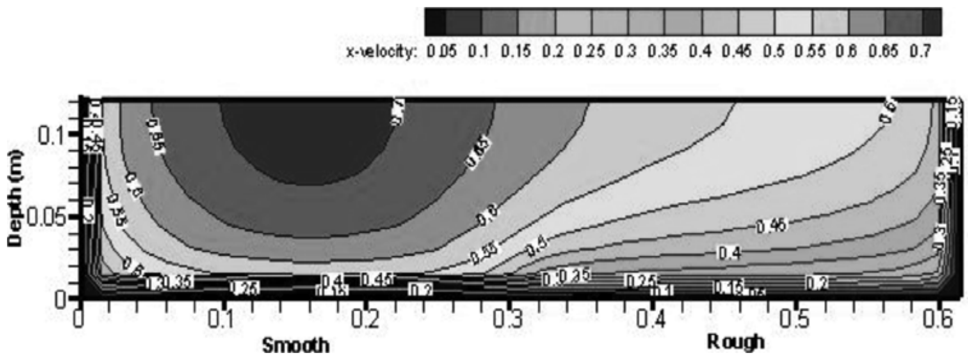

FIG. 7(a). PRIMARY VELOCITY CONTROUS

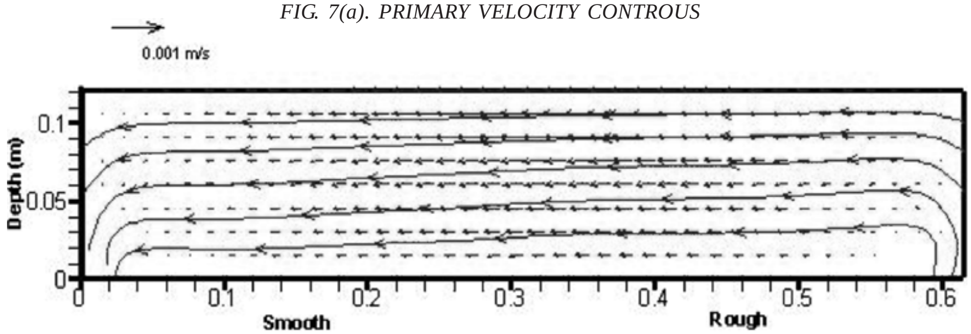

FIG. 7(b). SECONDARY VELOCITY VECTORS

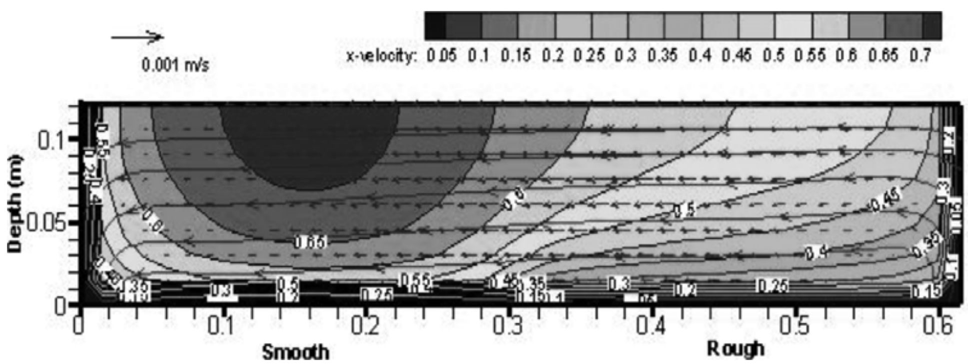

FIG. 7(c). PRIMARTY VELOCITY CONTROUS AND SECONDARY VELOCITY VECTORS

REFERENCES

[1] Jasson, M., Bridgeman, J., and Sterling, M., "Turbulent Flow Structure in a Heterogeneous Channel and their Effect on Conveyance Characteristics", Proceedings of Recent Research in Hydrology, Geology and Continum Mechanics, Volume 25, pp. 120-125, UK, 2010.

[2] Mclelland, S.J., Ashworth, P.J., Best, J.S., and Livesey J.R., "Turbulence and Secondary Flows Over Sediment Stripes in Weakely Bimodal Bed Material", ASCE Journal of Hydraulic Engineering, Volume 125, No. 5, pp. 463-473, 2000.

[3] Alhamdi, A.A.I., “Boundary Shear Stress and Velocity Distribution in Differentially Roughened Trapezoidal Open Channels”, Ph.D. Thesis, University of Birmingham, UK, 1991.

[4] Wang, Z.Q., and Chen, N.S., "Secondary Flow Over Artificial Bed Strips", Advances in Water Resources, Volume 28, pp. 441-450, 2005.

[5] Lauref, J., "The Structure of Turbulence in Fully Developed Pipe Flow", NACA Report, 1174, 1984.

[6] Souliotis, D., and Prinos, P., "Effect of a Vegetation Patch on Turbulent Channel Flow", Journal of Hydraulic Research, Volume 49, No. 2, pp. 157-167, Delft, Netherland, 2011.

[7] Jasson, M., Sterling, M., and Bridgeman, J., ''An Experimental Study of Turbulence in Heterogeneous Channel'', Proceedings of ICE Water Management, Volume 164, pp. 120-125, UK, 2012.

[8] Vermass, D.A., Uijttewaal, W.S.J., and Hoitink, A.J.F., 'Effect of Heterogeneous Bed Roughness on the Conveyance Capacity of Floodplains", Proceedings of NCR Days, pp. 34-35, Netherland, 2007.

[9] User Guide Fluent Lebnon: FLUENT Incorporated Lebanon, New Hampshire, USA, 2011.

[10] Jasson, M., "The Effect of Heterogeneous Roughness on Conveyance Capacity and Application to the Shiono-Knight Method'', Ph.D. Thesis, University of Birmingham, UK, 2012

[11] User Guide Gambit Gnterra Resources Park 10 Cavendish Court, Lebanon: NH 03766.2010.