Numerical Investigation of Developing Velocity Distributions in Open Channel Flows

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

RECEIVED ON 30.12.2012 ACCEPTED ON 20.03.2013

ABSTRACT

The velocity profiles in open channel flows start developing after entering into the channel for quite some length. All types of laboratory experiments for open channel flows are carried out in the fully developed flow regions which exist at some length downstream the inlet. In this research work an attempt has been made to investigate the impact of roughness and slope of the channel bed on the length required for establishment of fully developed flow in an open channel. A range of different roughness values along with various slopes were considered for this purpose. It was observed that an increase in roughness results in reduction of development length; and development length reduces drastically when roughness reaches to the range normally encountered in open channel flows with emergent vegetation or natural river flows. However, it was observed that the change of slope did not have any noticeable effect on development length. This work suggests that CFD (Computational Fluid Dynamics) technique can be used for getting a reliable development length before performing an experimental work.

Key Words: **Fully Developed Flow, Roughness, Computational Fluid Dynamics, Primary Velocity Contours, Boundary Conditions.**

1. INTRODUCTION

The knowledge of the length required for establishment of fully developed flow is the most important requirement for a detailed study of various aspects of three dimensional open channel flows. To the knowledge of the author, there is not too much research work available in literature in this area of open channel flows. The experiments being conducted now a days are still without a define guide in this regard.

There are always steep velocity gradients at the inlet and near the channel bed and walls. This is because of no slip boundary condition which exists between the fluid and solid walls. This no slip condition is a result of shear stress existing between the fluid and solid particles. This results in the formation of the boundary layer in which there are viscous forces dominant close to the bed. As the flow moves down stream this boundary layer starts growing towards the free surface of the channel and ultimately it reaches the free surface. The distance travelled by the water to reach this situation is called entrance length and from this length onward the flow profiles strictly remain unchanged along the length of the channel. It is very

* Assistant Professor, Department of Civil Engineering, University of Engineering & Technology, Taxila.

** Senior Engineer, Atomic Energy Commission, Islamabad.

*** Associate Professor, Department of Architecture Engineering & Design, University of Engineering & Technology, Lahore.

important to conduct all types of laboratory and numerical experiments in the fully developed region.

It has been established that for a pipe flow the length required for achieving a fully developed flow is around 50 times its diameter [1]. Keeping in mind this thing the flow development length for open channel flows will be around 200 times of its hydraulic radius or depth of the flow. However this seems to be an erroneous length as the flow develops in open channels much before this length. Some of the previous research made on this area was that of Raju, et. al. [2], Lien, et. al. [3], and Byrne, et. al. [4]. Some of them expressed the development length in terms of flow depth. For example, Lien et. al. [3] concluded that development length ranged from 30-150 times the depth of the flow. Similarly few researchers used smooth channel surfaces in their research work. However, still this is an ongoing research area because no definite criterion has been developed so far. Some times in numerical experiments researchers had to use a quite long channel to get a fully developed flow [5-6]. Most of the previous work on this area has been done experimentally as mentioned above and numerical tool has not been exploited too much to explore the entrance length required for flow establishment in open channel flows.

In this work, a CFD technique has been used for studying the development length in an open channel flow under different conditions. The three dimensional CFD code FLUENT [7] has been used to achieve this goal. It is a finite volume based code which employs Reynolds Averaged Navior-Stokes equations for solving the fluid flow problems. The mesh generation has been done through Gambit 2.3 [8]. The impact of bed roughness and slope has been observed on the development length of the channel. The primary velocity contours have been shown over various cross sections along the stream-wise direction under these changing conditions to get the location of the fully developed flow phenomenon. In each case the primary velocity profiles were also plotted over the verticals at mid section to get an idea of flow development from these profiles.

2. EXPERIMENTAL DATA FOR VALIDATION PURPOSES

The experimental data of Tominaga, et. al. [9] has been used for validation purposes. Tominaga et. al. performed a series of experiments on rectangular and trapezoidal channels with different hydraulic conditions. These experiments were carried out in a 12.5m long channel. It had x-sectional dimensions of 40x40cm. The bed of the channel consisted of painted iron plate and side walls were glass. The data was collected using hot film anemometer. In this simulation work only the case no. S1 for smooth rectangular channel has been utilized for validation purposes. For this particular case the flow depth is 5cm. This case has been used as a base case. The data used has been shown in Table 1.

3. NUMERICAL MODEL SETUP

In this research work three dimensional numerical code FLUENT has been used. It is a finite volume based code and solves three dimensional Navior-Stokes continuity and momentum equations for open channel flow problems [10-11]. The boundary conditions used included uniform velocity inflow at the inlet, pressure at the outlet, no slip wall boundary conditions on the bed and side walls, and symmetry boundary condition on the free surface. Gambit 2.3 has been used for the mesh generation process in this simulation work. A structured grid consisting of hexahedral elements was used in the mesh. The plan view of the mesh has been shown in Fig. 1.

TABLE 1. EXPERIMENTAL DATA USED IN THIS WORK

Case No.	Shape	Length (m)	Cross Section	Roughness Height (m)	Slope	Discharge (m^3/s)
S1	Rectangular	12.5	40x40cm	Smooth (0.000003)	0.000937 (1/1067)	0.00795

The default under relaxation factors were kept unchanged. The quadratic pressure-strain (SSG) type of Reynolds stress model has been used in this simulation work. The near wall treatment was achieved using standard wall function. Second order upwind schemes have been employed for spatial discretization of pressure, momentum, turbulent kinetic energy, its dissipation rate and Reynolds stress terms. Pressure velocity coupling has been achieved by utilising SIMPLE (Semi Implicit Method for Pressure-Linked Equation of Consistency) algorithm. The convergence criterion was set as $1x10^{-06}$.

After this setup, the model was validated using the experimental data shown in Table 1. The numerical results matched very closely to that of experimental ones. A mesh dependency test was then carried out by refining the mesh over the section which showed that mesh refinement has no improvement in the results. It proved that this model can be used with existing mesh for further systematic changes in roughness and slope to study their impact on flow development.

4. RESULTS AND DISCUSSIONS

The results for primary velocity contours and profiles for the base flow case of Tominaga et. al. have been shown in Fig. 2 (a-h). A number of cross sections and vertical lines at the mid section of each of these cross sections have been taken to investigate the flow development along the channel.

In the following diagrams 1, 2, 3 mean cross sections at 1, 2 and 3 meter distances down-stream the inlet respectively while line 1, 2 and 3 mean the mid section vertical lines over these cross sections. Table 2 shows various roughness and slope values for different cases investigated in this work.

4.1 Case-1

In Fig. 2 the primary velocity contours have been shown on different planes for the base Case-1. For the sake of clarity, a very close look has been given in each diagram.

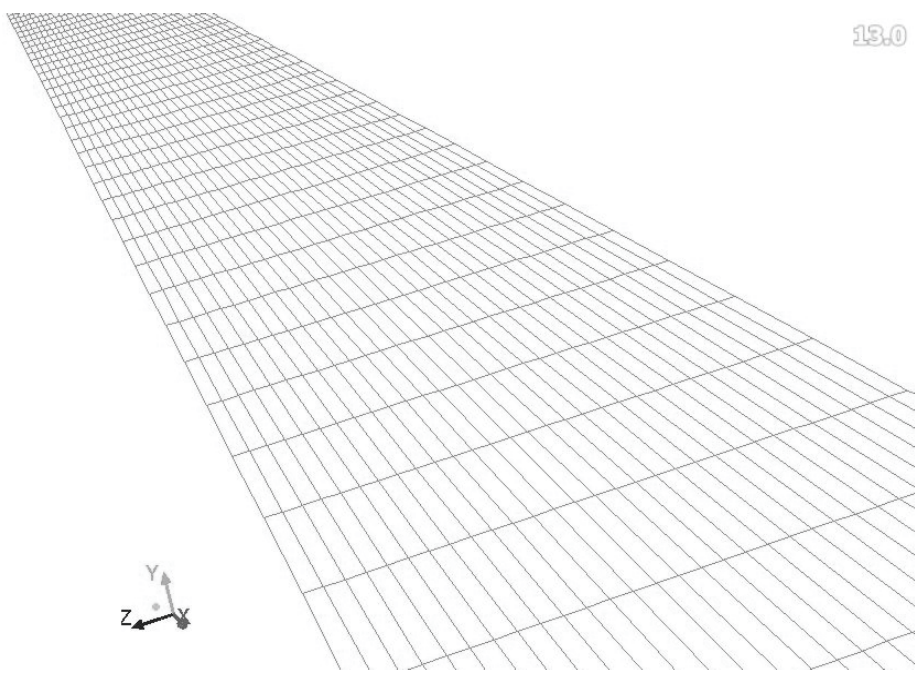

FIG. 1. PLAN VIEW OF THE MESH

A number of figures have been drawn to show the gradual development along the span. For example, the Fig. 2(a) shows velocity contours at the inlet and Section 1. This diagram indicates that inlet is fully covered by a constant velocity of 0.3973 m/sec which is inflow. But while reaching Section 1, the velocity value has risen to 0.43 m/sec (as shown by the legend on left of the diagram) and some velocity contours have developed over this Section 1. The Fig. 2(b) shows the velocity contours over inlet, Sections 1 and 2. It shows that velocity has now developed to 0.449 m/sec. The Fig. 2(c) shows the velocity contours over Sections 2, 3, 4 and 5. It shows that now the velocity contours have attained a reasonable shape but still there is a change between the successive sections (as the value on the legend is being changed and has not attained a constant value) and flow is still under developing process.

It shows that now the velocity has developed to 0.463 m/sec. The Fig. 2(d) shows the velocity contours over Sections 4, 5, 6, 7 and 8. The change in velocity contours shape is not clear from section 6 onward, however the legend of this contour diagram indicates a change in velocity value. The Fig. 2(e) shows the velocity contours over Sections 9, 10, 11, 12 and 12.5 (which is the outlet of the geometry). The legend value and velocity contours of

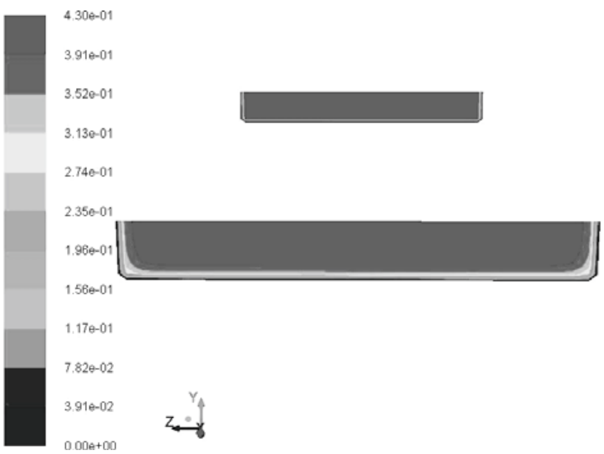

FIG. 2(a). PRIMARY VELOCITY CONTOURS OVER INLET AND SECTION 1

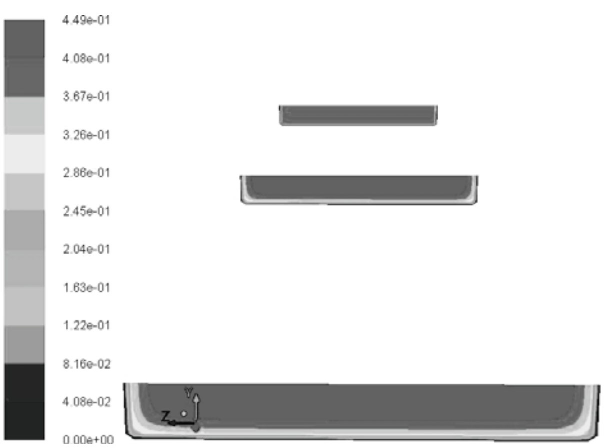

FIG. 2(b). PRIMARY VELOCITY CONTOURS OVER INLET, 1-2

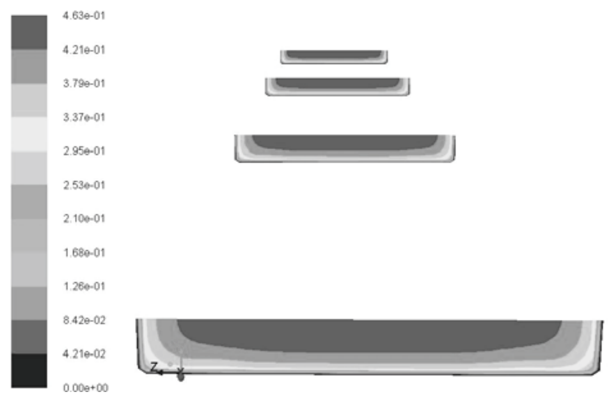

FIG. 2(c). PRIMARY VELOCITY CONTOURS OVER SECTIONS 2-5

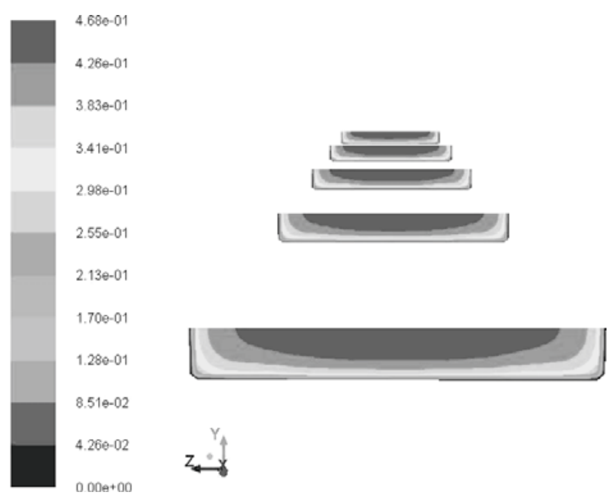

FIG. 2(d). PRIMARY VELOCITY CONTOURS OVER SECTIONS 4-8

Fig. 2(e) show that results are same as those at section 8. It means that it can be concluded that flow was fully developed at a distance 8 m down stream from inlet. In all these diagrams, the maximum velocity values occur at the free surface which can be attributed to the symmetric boundary condition taken at the free surface.

Similarly this flow development was investigated by sketching the primary velocity profiles over the depth of the vertical lines taken at the mid of these sections. This has been shown in Fig. 2 (f-h). The Fig 2(f) shows that velocity profiles are changing significantly from vertical lines 1-4. Similarly Fig. 2(g) gives changing velocity profiles from lines 4-8. Both these diagrams are indicating that velocities are constantly changing from sections 1 to 8. However Fig. 2(h) indicates that these velocity profiles remain same from line 9-12.

4.2 Case-2

After having a detailed description for the base case, only the most relevant diagrams have been presented here for the other cases to reach some conclusions. Fig. 3(a-f) represents Case-2 as described in Table 2. In Case-2, the roughness has been changed while keeping the slope constant. The roughness change

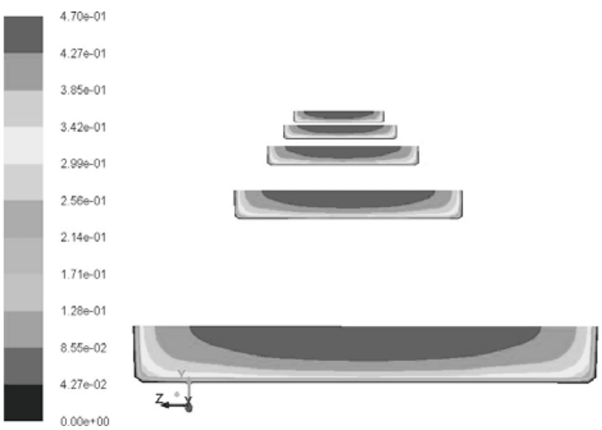

FIG. 2(e). PRIMARY VELOCITY CONTOURS OVER SECTIONS 9-12.5

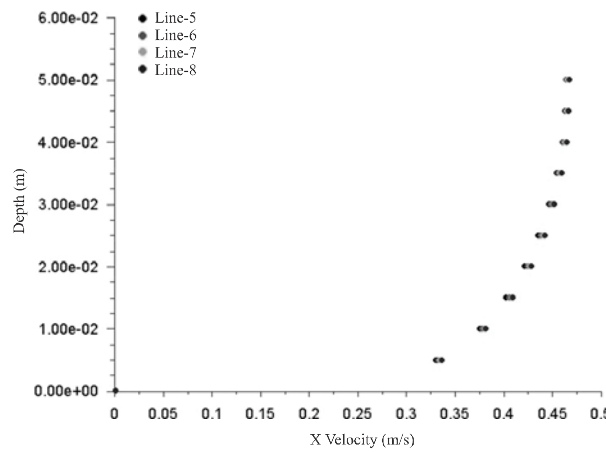

FIG. 2(g). VELOCITY PROFILES OVER VERTICAL 5-8

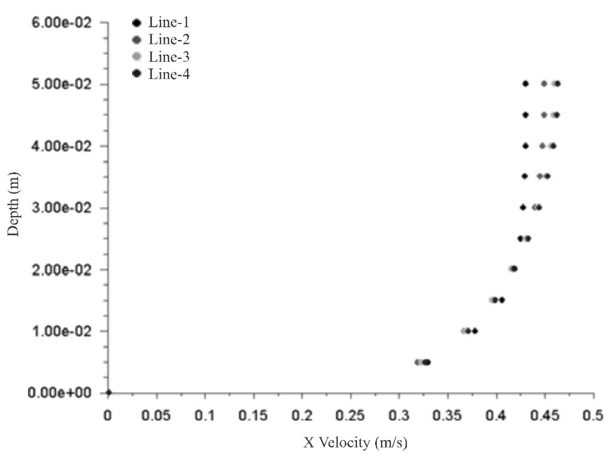

FIG. 2(f). VELOCITY PROFILES OVER VERTICAL 1-4

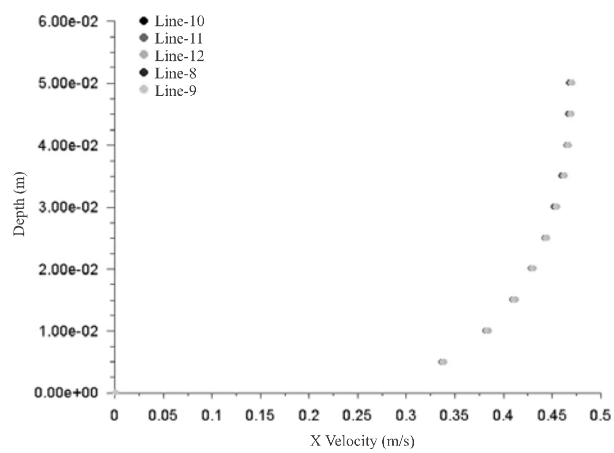

FIG. 2(h). VELOCITY PROFILES AT SECTIONS 8-12 results in an increase in wall friction. It impacts the velocity values minutely as shown in the Fig. 3(a-f). After having a look into these diagrams, it can be concluded that the velocity contours and profiles were fully developed after a length of around 7 meters from inlet. It means that increase in roughness results in an early development of flow and reduction in development length.

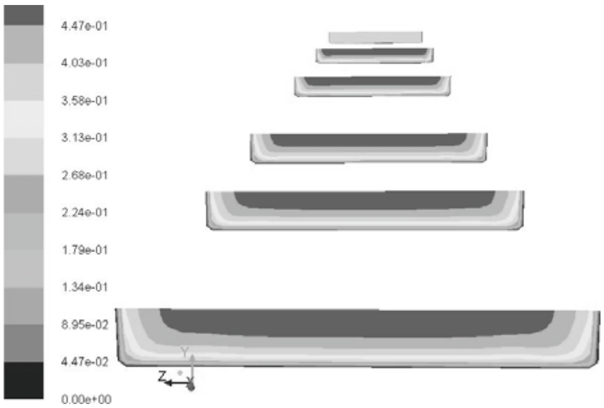

FIG. 3(a). PRIMARY VELOCITY CONTOURS OVER INLET, 1-5

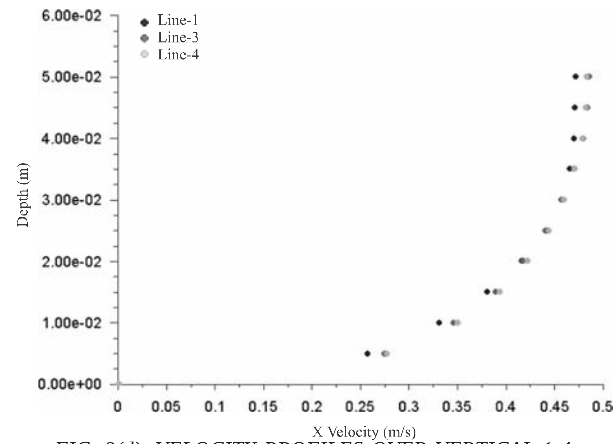

FIG. 3(d). VELOCITY PROFILES OVER VERTICAL 1-4

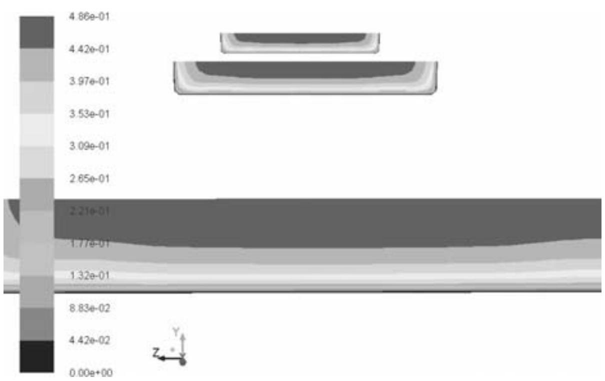

FIG. 3(b). PRIMARY VELOCITY CONTOURS OVER 5-7

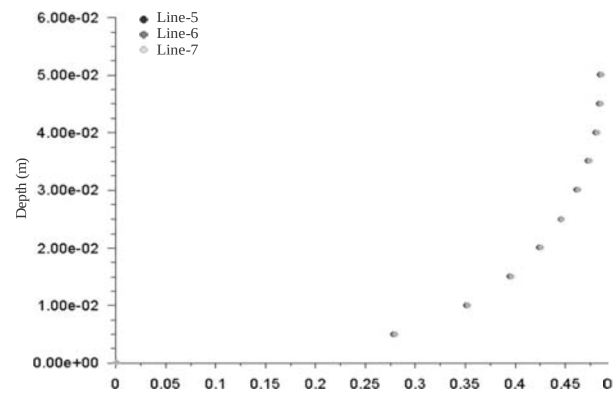

FIG. 3(e). VELOCITY PROFILES OVER VERTICAL 5-7

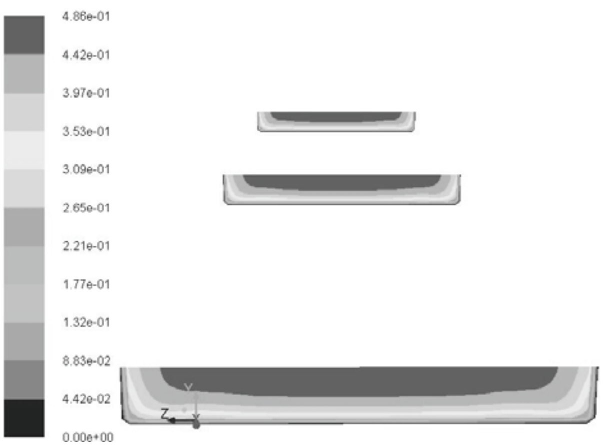

FIG. 3(c). PRIMARY VELOCITY CONTOURS OVER SECTIONS 7-9

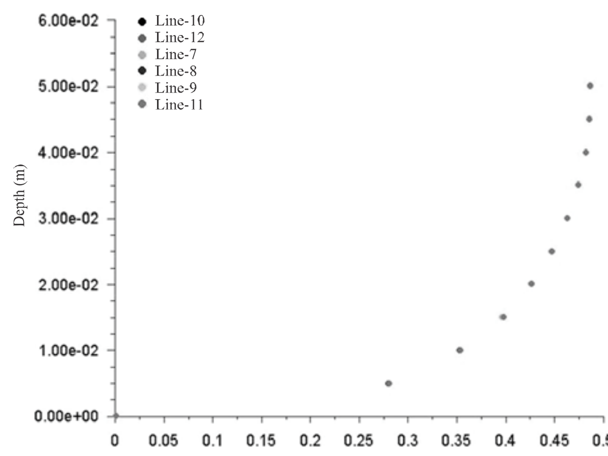

FIG. 3(f). VELOCITY PROFILES AT SECTIONS 7-12

4.3 Case 3-5

The results for Cases 3-5 have been shown in Figs. 4-6. In case of considerable increment in roughness such as Cases 4-5, as shown in Figs. 5-6, it has been observed both from velocity profiles and contour plots that flow establishment length reduces a lot i.e. even to an extent of 1.5-2.5m from the inlet. For Case-3, which has been shown in Fig. 4, the velocity profiles and contours seemed to have been developed upon reaching a length of 5m. The Fig. 4(f) is a clear indication of it. The contour plots of this case also suggest that the development length is around 5m from the inlet.

For Case-4, the approximate development length seems to be 3m as there is no change in velocity values after this length as is clear from Fig. 5(c and e). Although Fig. 5(e) shows some difference between profiles of line 3 and remaining lines but as far as the higher depths are

TABLE 2. VARIOUS CASES CONSIDERED IN THIS WORK

No.	Case No.	Shape	Roughness Height (m)	Slope
1.	Base Case, Tominaga et. al. Case S1)	Rectangular	Smooth (0.000003)	0.000937 (1/1067)
2.	Roughness Changed	Rectangular	0.005m	Constant Slope 0.000937
3.	Roughness Changed	Rectangular	0.01m	Constant Slope 0.000937
4.	Roughness Changed	Rectangular	0.05m	Constant slope 0.000937
5.	Roughness Changed	Rectangular	0.1m	Constant Slope 0.000937
6.	Slope Changed	Rectangular	Smooth (0.000003)	0.00125 (1/800)
7.	Slope Chared	Rectangular	Smooth (0.000003)	0.00150 (1/667)
8.	Slope Changed	Rectangular	Smooth (0.000003)	0.01 (1/100)

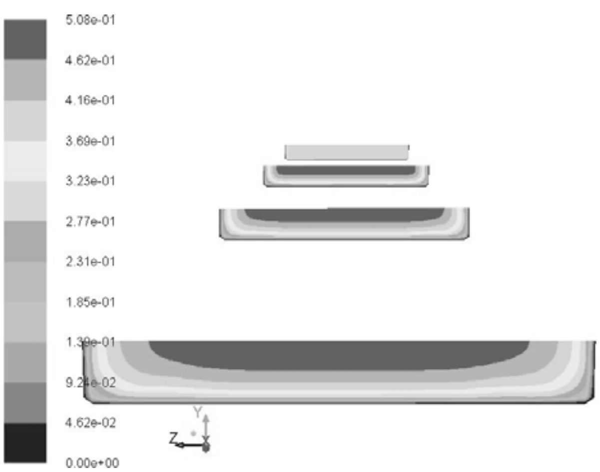

FIG. 4(a). PRIMARY VELOCITY CONTOURS OVER INLET, 1-3

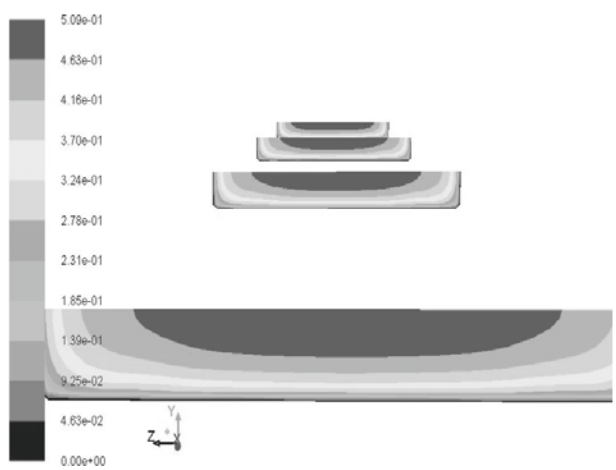

FIG. 4(b). PRIMARY VELOCITY CONTOURS OVER 3-6 concerned, this difference is less than 1%. Therefore, it can be assumed that velocity profiles have fully developed at this length. The Fig. 6 shows the flow case of maximum roughness i.e. the Case-5. As there is no change in velocity

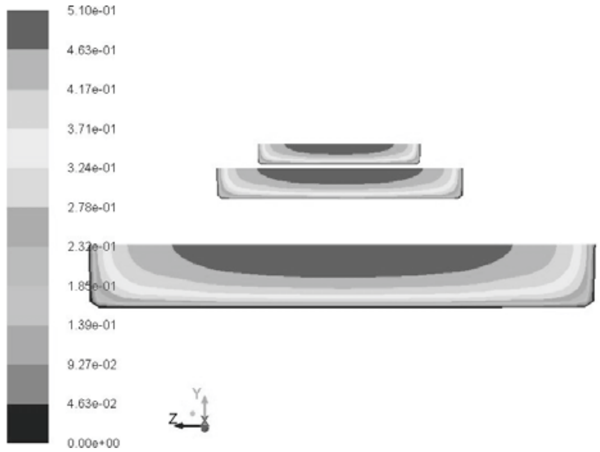

FIG. 4(c). PRIMARY VELOCITY CONTOURS OVER SECTIONS 6-8

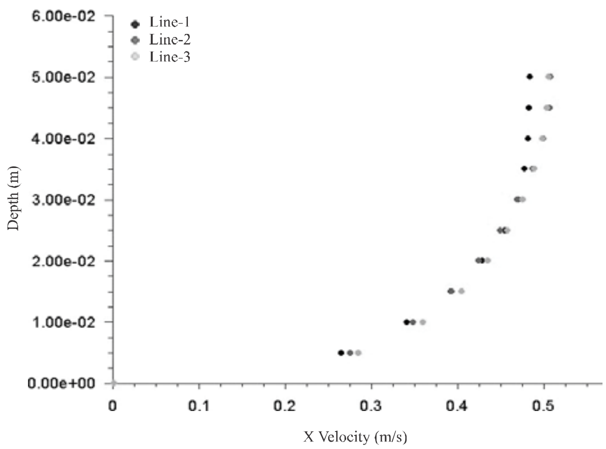

FIG. 4(d). VELOCITY PROFILES OVER VERTICALS 1-3

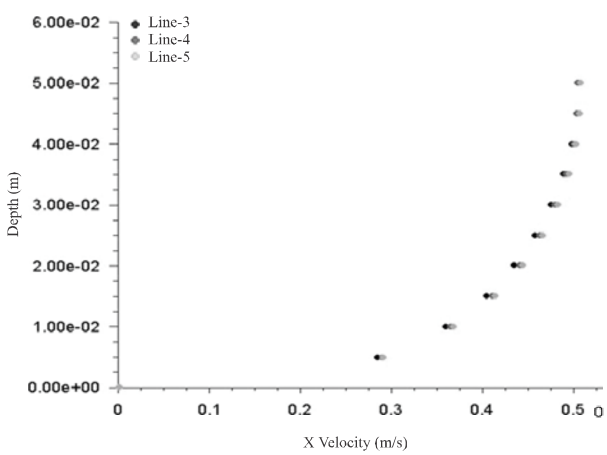

FIG. 4(e). VELOCITY PROFILES OVER VERTICALS 3-5

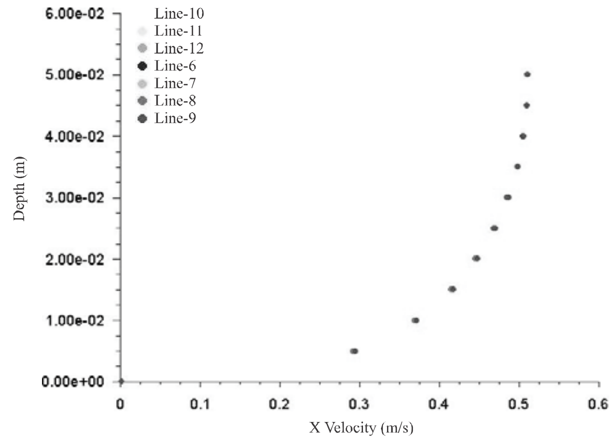

FIG. 4(f). VELOCITY PROFILES AT VERTICALS 5-12

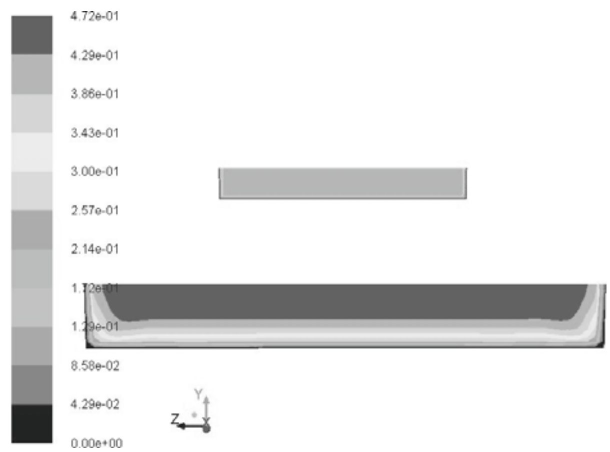

FIG. 5(a). PRIMARY VELOCITY CONTOURS OVER INLET AND SECTION 1

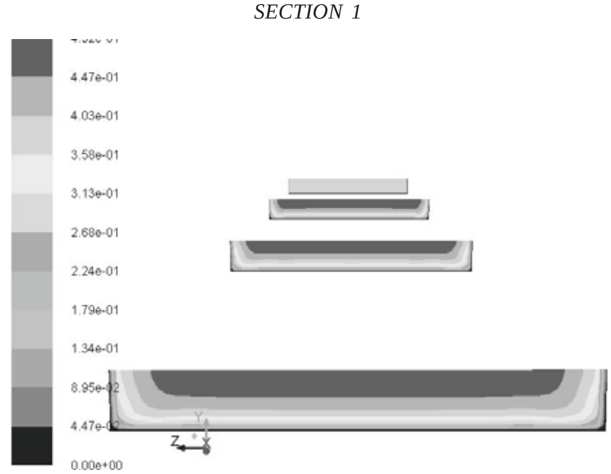

FIG. 5(b). PRIMARY VELOCITY CONTOURS OVER INLET, 1-3 contour values shown in Fig. 6(b-c), it suggests that the increased roughness has further reduced the development length to around 2m. Such a roughness is possible in a number of practical engineering application cases such as the presence of vegetation, natural rivers, etc.

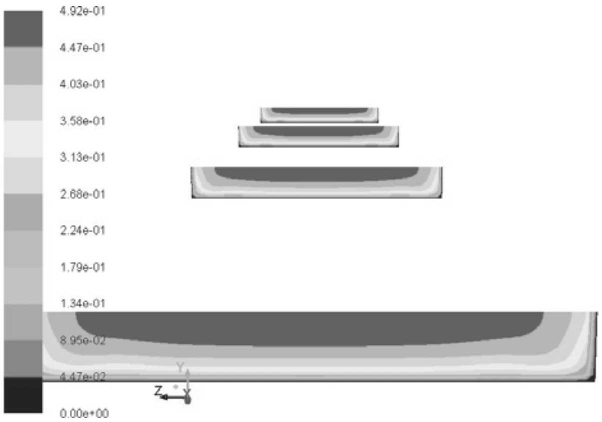

FIG. 5(c). PRIMARY VELOCITY CONTOURS OVER 3-6

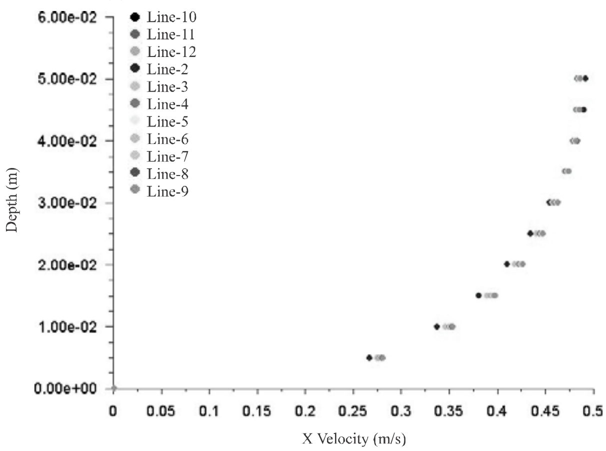

FIG. 5(d). VELOCITY PROFILES AT (VERTICALS IN THIS FIG. AND ALSO IN FIG. 5(c)

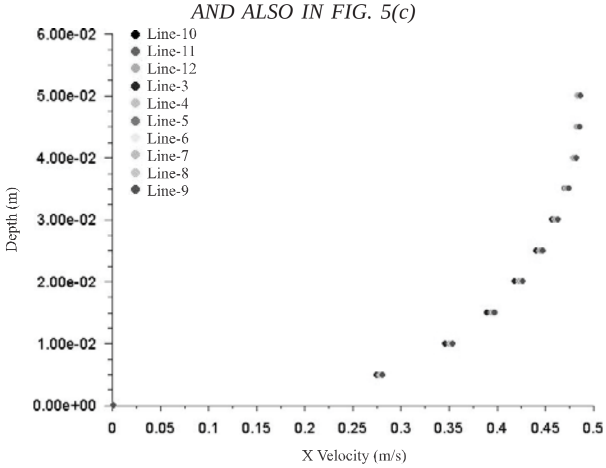

FIG. 5(e). VELOCITY PROFILES AT (VERTICALS IN THIS FIG. AND ALSO IN FIG. 5(c)

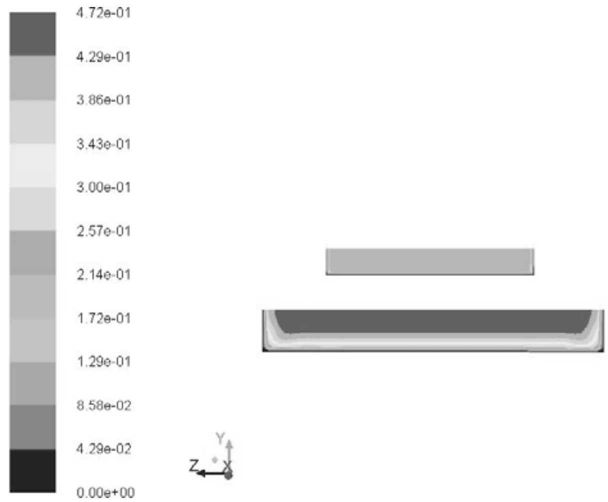

FIG. 6(a). PRIMARY VELOCITY CONTOURS OVER INLET, 1

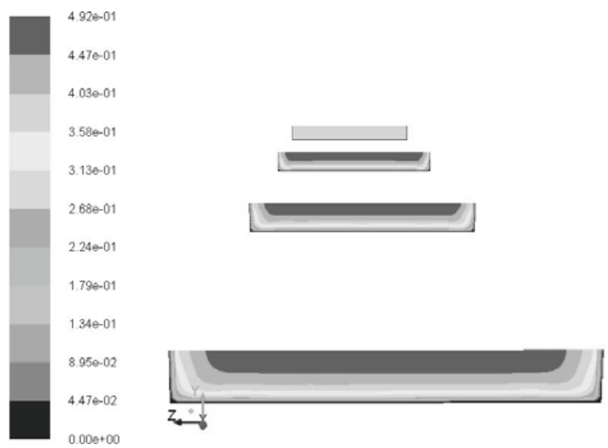

FIG. 6(b). PRIMARY VELOCITY CONTOURS OVER SECTIONS INLET, 1-3

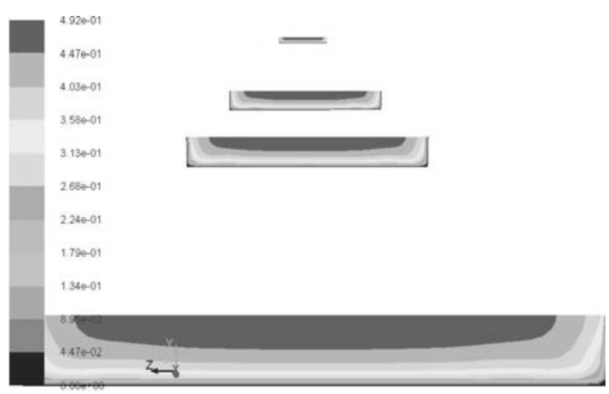

FIG. 6(c). PRIMARY VELOCITY CONTOURS OVER 2,8,9 AND10

As far as the change in slope is concerned, its impact on developing velocity distributions has been demonstrated in Figs. 7-9. These diagrams correspond to Cases 6-8 as given in Table 2. All these diagrams have shown almost similar results, i.e. no change in development length has been observed in all the cases. The flow has been seen to be developed after 8m of length downstream the inflow. For example Fig. 7(a-c), which represents Case-6, shows that the velocity is continuously changing till the Section 8. The same is the situation for Case-7; in this case the Fig. 8(f) clearly shows that the velocity profiles are exactly identical from lines 8 onward whereas they keep on changing from inlet to Section 8. Similar is the situation for Case-8 as shown in Fig. 9. If there is some difference in flow development length from one case of the slope change to the other, then it is less than 2-3%. It can be concluded that the bed slope has no contribution towards the development length.

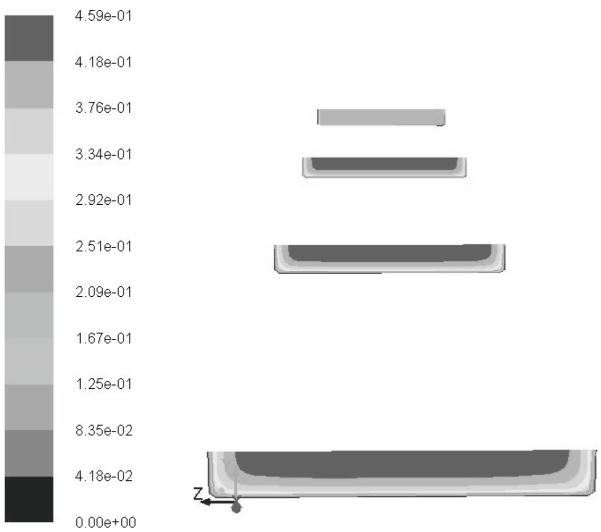

FIG. 7(a). PRIMARY VELOCITY CONTOURS OVER INLET, 1-3

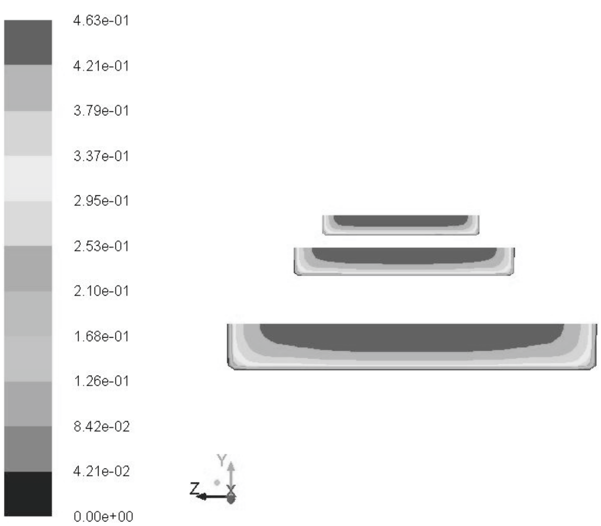

FIG. 7(b). PRIMARY VELOCITY CONTOURS OVER 3-5

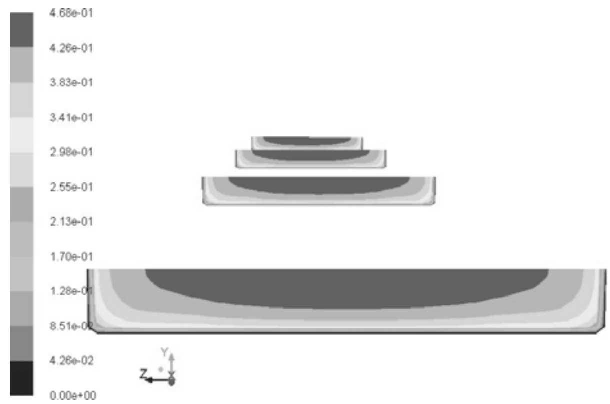

FIG. 7(c). PRIMARY VELOCITY CONTOURS OVER 6-9

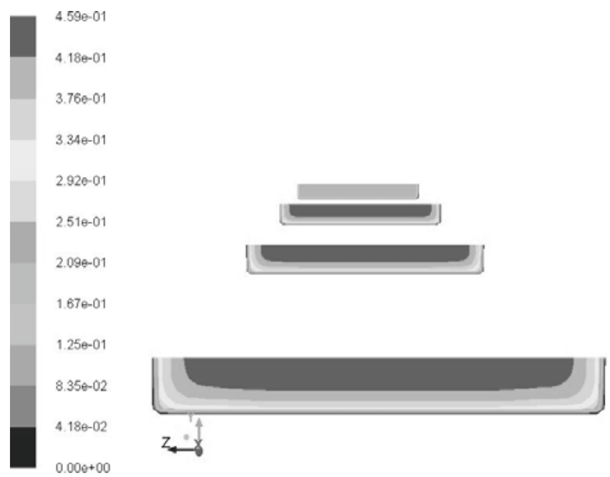

FIG. 8(a). PRIMARY VELOCITY CONTOURS OVER INLET, 1-3

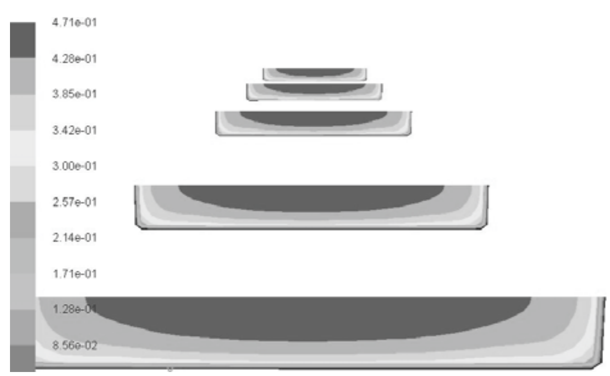

FIG. 8(b). PRIMARY VELOCITY CONTOURS OVER 4-8

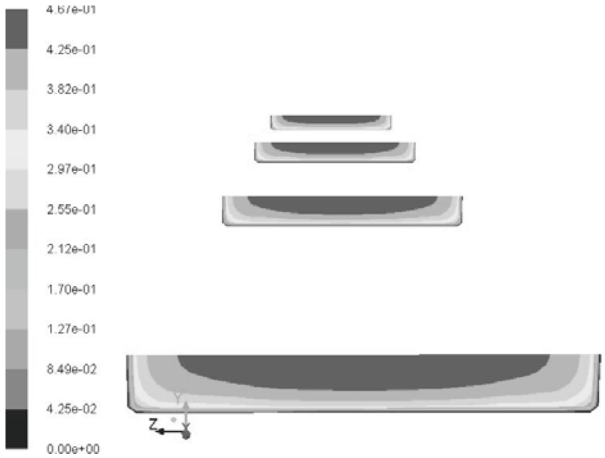

FIG. 8(c). PRIMARY VELOCITY CONTOURS OVER 9-12 OUTLET

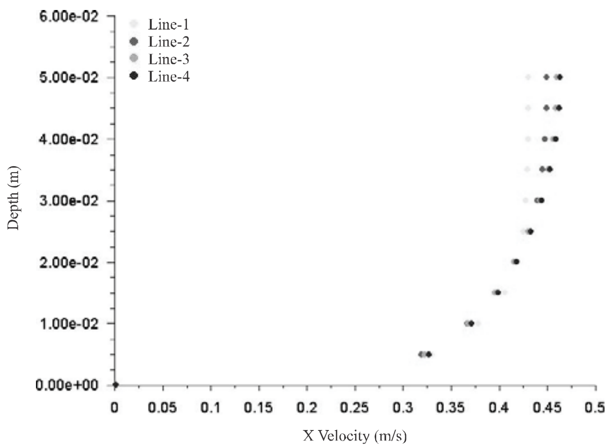

FIG. 8(d). VELOCITY PROFILES OVER VERTICAL 1-4

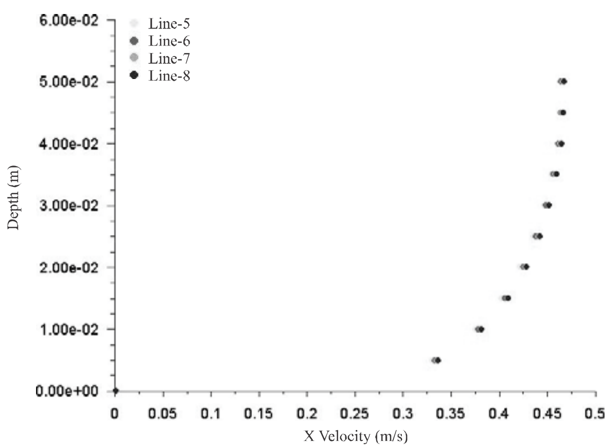

FIG. 8(e). VELOCITY PROFILES OVER VERTICAL 5-8

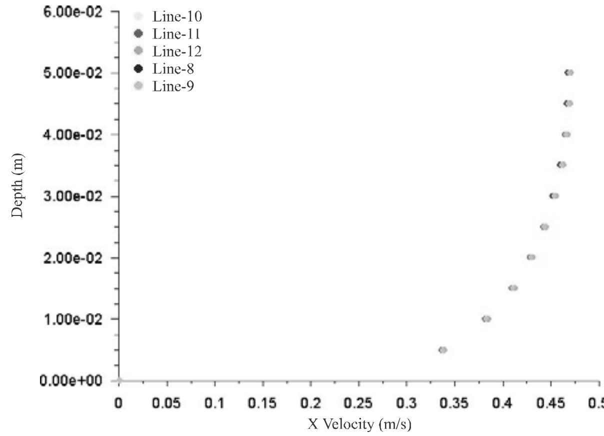

FIG. 8(f). VELOCITY PROFILES AT VERTICALS 8-12

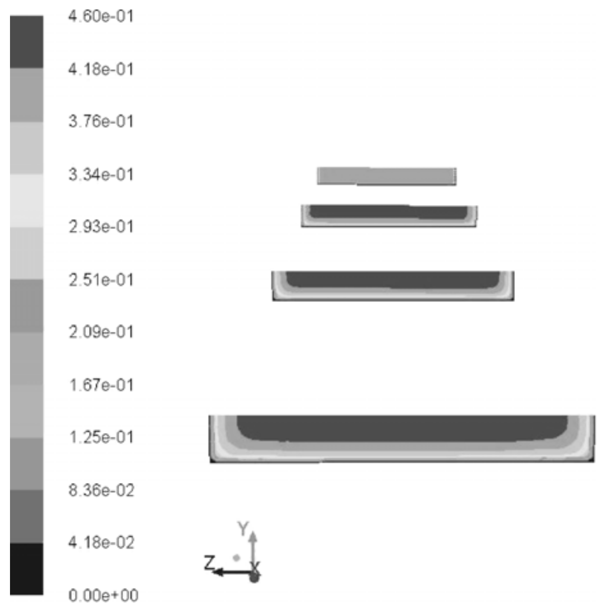

FIG. 9(a). PRIMARY VELOCITY CONTOURS ON INLET, 1-3

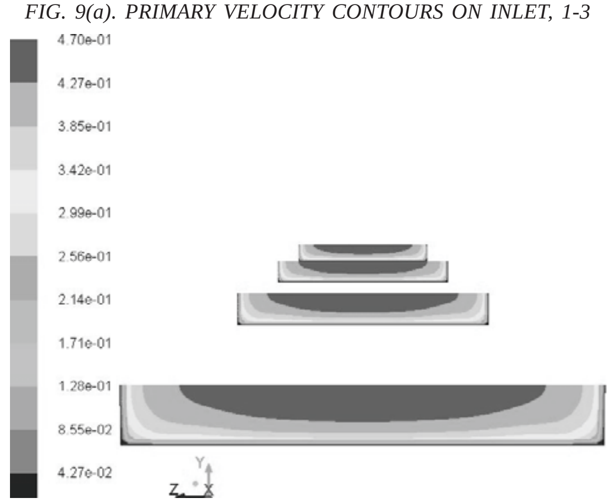

FIG. 9(b). PRIMARY VELOCITY CONTOURS ON 6-9

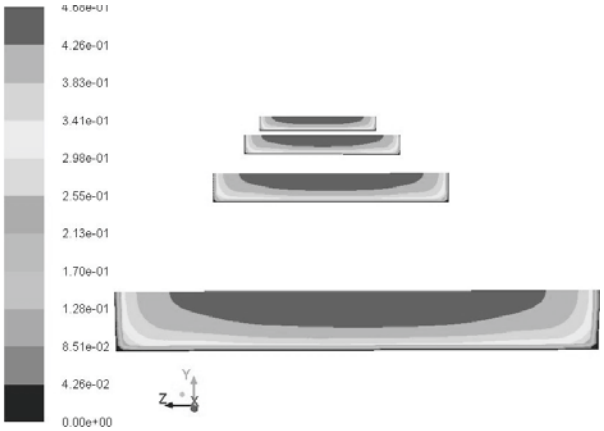

FIG. 9(c). PRIMARY VELOCITY CONTOURS ON 9-12

5. CONCLUSIONS

A CFD based numerical study has been done for exploring the impact of bed roughness and channel slope on the flow development length. For this purpose the primary velocity contours and velocity profiles were considered at successive sections along the channel. It was observed that increase in roughness results in a reduction of development length. This reduction is minor when roughness increase is very small, however for high roughness values there is a considerable reduction in development length. On the other hand it was observed that the channel bed slope has negligible impact on the length of the flow development. From this research work, it can be concluded that CFD can be used as an effective tool for obtaining flow establishment length before performing experimental work in the laboratory.

ACKNOWLEDGEMENTS

The authors are highly acknowledged to Higher Education Commission, Pakistan for providing CFD facilities at University of Engineering & Technology, Taxila, and Lahore, Pakistan. All this helped them in conducting this research work.

REFERENCES

[1] Shaughnessy, E.J., Katz, I.M., and Schaffer, J.P., "Introduction to Fluid Mechanics", Oxford University Press, pp. 791-793, UK, 2005.

[2] Raju, K.G.R., Asawa, G.L., and Mishra, H.K., "Flow-Establishment Length in Rectangular Channels and Ducts", ASCE Journal of Hydraulic Engineering, Volume 126, No. 7, pp. 533-539, USA, 2000.

[3] Lien, K., Monty, J.P., Chong, M.S., and Ooi, A., "Entrance Length for Fully Developed Turbulent Channel Flows", 15th Australian Fluid Mechanics Conference, Volume 15, pp. 356-363, Australia, December, 2004.

[4] Byrne, J., Hatton, A., and Marriott, P., "Turbulent Flow and Heat Flow in the Entrance Region of a Parallel wall Passage", Proceedings of Institute of Mechanical Engineers, Volume 184, pp. 697-712, 2007.

[5] Stoven, V.R., Grim, J.P., Buxton, A.P., and Tait, S.J., "Parametric Studies on CFD Models of Sewerage Structures", 9th International Conference on Urban Drainage, Volume 9, pp. 280-289, Portland, 2002.

[6] Han, S.S., Biron, P.M., and Ramamurthy, A.S., "Three-Dimensional Modelling of Flow in Sharp Open Channel Bends with Vanes". IAHR Journal of Hydraulic Research, Volume 49, No. 1, pp. 64-72, Netherland, January, 2011.

[7] Fluent, "User Guide Manual", FLUENT Incorporated, New Hamshire, USA, 2011.

[8] Gambit 2.3, "User Guide Manual', Gnterra Resources Park 10 Cavendish Court, New Hamshire, USA, 2010.

[9] Tominaga, A., Nezu, I., Ezaki, K., and Nakagawa, H., "Three Dimensional Turbulent Structure in Straight Open Channel Flow", IAHR Journal of Hydraulic Research, Volume 27, No. 1, pp. 149-173, Netherland, January, 1989.

[10] Rodi, W., "Turbulence Models and Their Applications in Hydraulics, A State of the art Review", IAHR Monograph, Netherland, 1980.

[11] Nezu, I., and Nakagawa, H., "Turbulence in Open Channels", IAHR Monograph, Netherland, 1993.