Journal of Applied Research in Water and Wastewater 2 (2014) 53-58

Razi University

Journal of Applied Research in Water & Wastewater

Journal homepage: www.arww.razi.ac.ir

Original paper

Developing turbulent flows in rectangular channels: A parametric study

Hossein Bonakdari[1,*], Gislain Lipeme-Kouyi[2], Girdhari Lal Asawa[3]

[1]*Department of Civil Engineering, Razi University, Kermanshah, Iran.*
[2]*Department of Civil and Environmental Engineering, University of Lyon, Villeurbanne, France.*
[3]*Department of Civil Engineering, GLA University, Mathura, India.*

ARTICLE INFO

Article history:
Received 04 January 2014
Received in revised form 27 March 2014
Accepted 16 April 2014
Available online 28 June 2014

Keywords:
Fully developed flow
Establishment length
Open channel flow
Numerical modelling
Velocity field

ABSTRACT

The developing turbulent flow in an open channel is a complex three-dimensional flow influenced by the secondary currents and free surface effects and is, therefore, not amenable to analytical solution. This paper aims to study the impact of three key hydraulic parameters (relative roughness, the Froude number and the Reynolds number) on the establishment length using computational fluid dynamic (CFD) analysis. CFD analysis is based on the use of the ANSYS-CFX commercial code. The CFD strategy of modelling is validated against experimental velocity distribution in a cross-section and a good agreement is achieved. A dimensionless length is suggested for predicting the length of the developing flow zone for rectangular open channel. A linear relationship has also been developed for assessing the establishment length.

Nomenclature

g	gravitational acceleration	(m/s^2)
Le	establishment length	(m)
H	flow depth	(m)
P	pressure	(Pa)
u_*	shear velocity	(m/s)
U	flow velocity	(m/s)
Umax	maximum velocity	(m/s)
y	width	(m)
z	height	(m)
Z	vertical elevation	(m)
ζ	channel roughness	(m)
δ	boundary layer thickness	(m)
ρ	density of fluid	(kg/m^3)
ν	kinematic viscosity	(m^2/s)
$\overline{u_i u_j}$	Reynolds stress tensor	(Pa)
F	Froude number	
R	Reynolds number	
Iec	non dimensional index	

$z+= \frac{u_* z}{\nu}$

1. Introduction

Turbulent flows in ducts and open channels are often encountered in engineering. The most basic requirement for the experimental and numerical study of fully developed three-dimensional open channel flows is the knowledge of the channel length necessary for its establishment. Indeed, in open channels, the velocity gradients in the entrance and near the channel bed are high due to the growing boundary layer (see Fig. 1).

Corresponding author, E-mail: bonakdari@yahoo.com

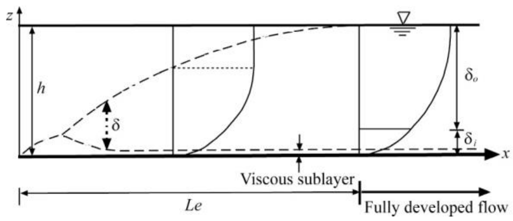


Fig. 1. Boundary layer and fully developed open channel flow.

The boundary layer thickness (δ) increases with distance from the entrance of the channel. After a certain distance (establishment length, Le), the boundary layer reaches the free surface where after the velocity profile remains invariant. Information about this establishment length is required for any experimental study and numerical modelling of free surface flows. Beyond this distance, the flow is fully developed. The turbulent structure of open channel flows in the developed zone can be divided into two sub-regions, (Nezu and Nakagawa 1993; Cebeci 2004): inner region (δ_i): z/h<0.2 and outer region (δ_o): 0.2≤z/h≤1.0.

Most experimental facilities for open channel flows are cited as being sufficiently long, yet no formal definition or documentary evidence of the fully developed condition is widely accepted. Determination of the establishment length (Le) under different hydraulic and geometric conditions is of vital importance in open channels as meaningful flow studies must be carried out in the developed flow zone. The establishment length or development length in an open channel has been studied in many ways over the years. While considerable amount of information is available regarding the development of flow in a circular pipe, there is limited information available in this respect for open channels to enable one to decide the length required for the establishment of fully developed turbulent flow in channels.

Nikuradse (1933) compared the mean velocity profiles at successive stream-wise lengths and concluded that the fully developed zone in a pipe was achieved in his experiments at a distance of fifty times the diameter from the entrance. Wang and Tullis (1974) reported similar results for the developing turbulent flow in a pipe. Since the hydraulic radius of a circular pipe is equal to one fourth of its diameter, the length required for establishment of the fully developed flow in an open channel can be taken as 200 times the hydraulic radius (or the depth of flow in a wide channel) using the above criterion. This length appears to be rather large in the context of lengths of the existing flumes in different laboratories.

Flow in a rectangular channel is characterized by the presence of corners and the free surface. Thus, the flow structure in any developing turbulent flow in channels is different from that in circular pipes. The secondary currents are induced mainly due to the inequality of the normal turbulent stresses at different locations in the cross-sectional plane (Perkins 1970). These secondary currents affect the primary mean flow field and cause the maximum velocity to occur below the free surface (dip phenomenon). It is important to investigate their influence in the developing flow region of a channel. The developing flow is a complex three-dimensional flow influenced by the secondary currents and free surface effects. CFD modelling can, probably, be used to analyze data in the developing flow region of a rectangular open channel for different flow conditions and wall roughnesses.

In the present study, the experimental data obtained by Tominaga et al. (1989) were processed to validate CFD modelling strategy and, then, the validated numerical approach was used to study the length of flow establishment in open channels as a function of wall roughness (the same roughness value is considered for both the side walls and channel bed) and relevant flow parameters.

2. Establishment length in open channel

The studies (either numerical or experimental) on the flow should be made only in the fully developed zone. Indeed, upstream of this zone, the parameters in the vicinity of the entrance are under the influence of a boundary condition that is rather uncertain. The choice of a short channel length induces the risk of the test section not being in the developed zone. However, the choice of a very large length increases the experimental cost or enhances the computing time in numerical studies. Several studies have been carried out in the past on the prediction of the establishment length. From these studies, the establishment length Le in an open channel is seen to vary from 50h to about 150h. Here, h is the depth of flow in the channel. Table 1 recapitulates the results obtained from the previous studies.

Table 1. Summary of selected studies for prediction of the establishment length value.

Reference	Type of channel	Le
Grass (1971)	Smooth & rough	130h
Dean (1978)	Smooth	55h
Nezu and Rodi (1985)	Smooth	60h-90h
Cordaso et al. (1989)	Smooth	69h-108h
Graf (1991)	Rough	140h
Ranga Raju et al. (2000)	Smooth & rough	50h-100h
Zanoun et al. (2003)	Smooth	115h
Lien et al. (2004)	Smooth	130h

These previous investigations, however, do not allow quantitative and precise evaluation of the establishment length. Indeed, in these studies, the main parameter studied was the hydraulic radius (or the flow depth). There are some other parameters such as velocity, channel width, and channel roughness which too affect the establishment length. However, these were not considered in the above-mentioned studies. Kirkgoz and Ardichoglu (1997) have, from their experimental data in a smooth channel, found that the dimensionless length of the developing flow region (Le/h) is related to the Reynolds number (R) and Froude number (F) as follows (R/F<500000):

$$\frac{Le}{h} = 76 - 0.0001\frac{\mathrm{R}}{\mathrm{F}} \quad (1)$$

The measured velocity profiles of 12 tests of their investigation showed that the length of the boundary layer development varies between 50h to 70h. There is no clearly defined criterion for flow establishment length in channels. One could possibly define the length of establishment, Le, as the distance beyond which: (1) a characteristic boundary layer thickness like the displacement or momentum thickness becomes constant, or (2) the mean velocity profile along the centre line of the channel remains the same, or (3) the mean velocity profile along the whole of the cross section remains the same. There is evidence to suggest that the lengths obtained using these different criteria are not the same. In order to take into account the wall effect on the velocity field in the cross section, this study is based on criteria (3) to determine the establishment length.

3. Methodology
3.1. CFD modelling strategy

The present numerical study makes use of the Ansys - CFX software package for solving three-dimensional fundamental flow equations and the package enables one to calculate the velocity fields in any cross section.

3.1.1. Governing fluid flow equations

Two key equations for the fluid motion in an open channel are: (i) the law of conservation of mass for an incompressible fluid in Eulerian form,

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (2)$$

and (ii) the Reynolds' time-averaged Navier-Stokes equations for an incompressible turbulent fluid flow

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{g}\frac{\partial Z}{\partial x_i} - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial U_i}{\partial x_j} - \overline{u_i u_j}\right) \quad (3)$$

where x_i's, represent the coordinate axes, U_i's are the mean velocities in the x (stream-wise), y (lateral) and z (vertical) directions, Z is the vertical elevation, p is the pressure, ρ is the fluid density, and $\overline{u_i u_j}$ are the components of the Reynolds stress tensor.

3.1.2. Turbulence closure

Since most of the practical engineering flows are large Reynolds number flows, the simulation of turbulence is of great importance in order to obtain accurate numerical results. Before Eqs. 2-3 can be solved, a turbulence model must be introduced for determining the Reynolds stresses ($\overline{u_i u_j}$) appearing in the momentum equations, Eq. 3. The software provides various turbulence models. Stovin et al. (2002) have shown the influence of the turbulence model on the ability to represent the complexity of turbulent flows. Bonakdari (2006) and Bonakdari and Zinatizadeh (2011) have shown that the isotropic models are unable to represent the 3-D behavior of the velocity fields in narrow channels. Therefore, the anisotropic Reynolds Stress Model (RSM) has been used.

3.1.3. Boundary conditions

Boundary conditions around the solution domain are required in order to determine the solution of the governing fluid flow equations. The common boundary conditions in the open channel flow problems are:

(1) The water level at the inlet boundary needs to be given and this should be consistent with the mean velocity of flow in the channel.

(2) At the outlet, pressure is specified at the center of the cell face, while Cartesian velocity components and turbulence quantities are extrapolated from the interior of the cell using a second-order extrapolation.

(3) The interaction between the fluid and boundary walls is of great importance in the turbulent flows. Due to the strong velocity gradients occurring near the walls, a large amount of turbulence is generated there. This turbulence plays a very important role in several physical phenomena such as heat exchange and reattachment of the separated regions. For the present analysis, the wall function approach outlined by Launder and Spalding (1974) was used. This, in effect, means that the boundary conditions are not specified right at the wall but at a point, outside the viscous sub-layer, where the logarithmic law of the wall prevails.

(4) Assuming that the atmosphere exerts no shear and no inertia, equality of forces on the two sides of the interface is enforced. The free surface location is determined by the Volume of Fluid (VOF) method (Hirt and Nicholas 1981) which has been incorporated into the solution of the Reynolds' time-averaged Navier-Stokes equations in order to take into account the free surface effects on the Reynolds stresses and, thus, velocity distribution in the cross section. In the commercial package Ansys-CFX, the water flow and some of the air flow above the water are solved simultaneously. The governing fluid

Please cite this article as: H. Bonakdari, G. Lipeme-Kouyi, G. L. Asawa , Developing turbulent flows in rectangular channels: A parametric study, Journal of Applied Research in Water and Wastewater, 1(2), 2014, 53-58.

flow equations for air and water are expressed in a single form but with different physical properties. A second order implicit scheme is used to discretise the governing equations i.e., the continuity equation, the momentum equations, and the turbulence model.

3.1.4. Computational meshes

To design a computational mesh suitably spaced for the solution domain described previously, different structured meshes with various cell concentrations were tested (Bonakdari, 2006). A prismatic mesh with rectangular base is used because such a mesh can be conveniently generated for the geometry of the channel; it also gives the highest accuracy (Bonakdari, 2012). The cell density should be increased in the flow regions with large gradients in velocity and free surface profiles (Akoz et al., 2009). Relatively finer local meshes in the vertical direction were used particularly near the channel bed in the water zone so that the first mesh point remained within the viscosity-affected wall region for z+(= $\frac{u_* z}{\nu}$)≤30, where high-gradient velocity profiles occur. The mesh density is decreased in the air zone. In the water phase, this grid is composed of cells with rectangular sections with its smaller side in the vertical direction so as to correspond with a higher gradient of flow characteristics in this direction; in the air phase, stepping away from the water surface, compression decreases linearly. A value of 10-6 has been employed as the convergence criterion at each time step for the sum of each of the normalized residuals over the whole fluid domain for all the governing fluid flow equations and also the mass residuals. The maximum number of iterations was equal to 30000. However, if the convergence criterion is reached for all the residuals, the simulation was stopped before reaching 30000 iterations.

3.2. Experimental data

The experimental results obtained from the study of Tominaga et al. (1989) were used in the present study. The bed of the channel was a painted iron plate and the side-walls were of glass. The experimental study was carried out in smooth and rough rectangular open channels. In rough cases, the roughness elements were glass beads with l2 mm diameter and they were densely attached to the wall. The results were obtained in a channel with 12.5 m length, and a square cross section (0.40 m × 0.40 m). The experiments were performed with the Reynolds number equal to 1.9×104 and Froude number equal to 0.19 for a depth of flow equal to 0.10 m and the mean velocity equal to 0.187 m/s. Velocity measurements were performed using a hot-film anemometer. Velocities were measured at the mid-section at about 100 locations. In this case, the flow width to depth ratio is smaller than 5. In such hydraulic contexts, the maximum velocity is below the free surface (dip phenomenon) unlike the usual accepted situation for river flows (Stearns 1883, Nezu and Nakagawa 1993).

3.3. Methodology to assess the impact of the relative roughness, Reynolds number and Froude numbers

Dimensional analysis is a powerful tool for deriving the dimensionless relationships among the variables governing any fluid flow. The variables that affect the establishment length Le are the mean velocity in the stream-wise direction (U), depth of flow in the channel (h), channel roughness (ζ), mass density (ρ) and kinematic viscosity (ν) of the fluid, and gravitational acceleration (g). The dimensional analysis yields the following functional relation:

$$Le/h = f\left(\zeta/h,\ \frac{U}{\sqrt{gh}},\ \frac{Uh}{\nu}\right) \quad (4)$$

$$Le/h = f\left(\zeta/h,\ F,\ R\right) \quad (5)$$

To study the effect of variations of the relative roughness, Reynolds number, and Froude number, it is necessary to vary one while keeping the other dimensionless numbers constant.

4. Results

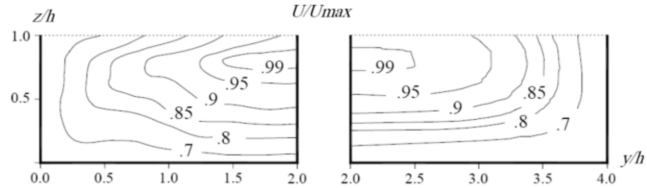
4.1. Experimental validation of the CFD modelling strategy based on experimental data

The validity of the proposed model was evaluated by the experimental results of Tominaga et al. (1989). For the simulated case, the average water depth is h = 0.1 m and the bulk mean velocity at the entrance is equal to 0.187 m/s corresponding to the mean velocity measured experimentally by Tominaga et al. (1989). This channel is narrow with aspect ratio (width/depth) less than 5. Therefore, the secondary currents move the fluid with relatively high stream-wise momentum towards the central portion of the channel and cause the observed depression of the maximum velocity below the free surface called the dip phenomenon (Nezu and Nakagawa, 1993). Computational Fluid Dynamics (CFD) modelling was able to simulate the iso-velocity field in the whole of the cross section. Figure 2 illustrates the comparison between the experimental data and numerical results. In order to quantitatively compare the performance of the numerical model in the entire cross section with the measured results, two indicators, relative error (Rerr) and root-mean-square error (RMSE) were used and which are as follows:

$$R_{err} = \left| \frac{U_{mo} - U_{exp}}{U_{exp}} \right| \quad (6)$$

$$RMSE = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(\frac{U_{mo} - U_{exp}}{U_{exp}}\right)^2} \quad (7)$$

where, n is the total point measurements in the cross section, U_{mo} the estimated velocity according to the numerical modelling and U_{exp} the measured velocity. The values of Rerr = 3% and RMSE = 4.5% obtained showed good agreement between the modelled and experimental data. CFD results show the dip phenomenon centered in the middle of the main channel at 70% of the water level from the bed.

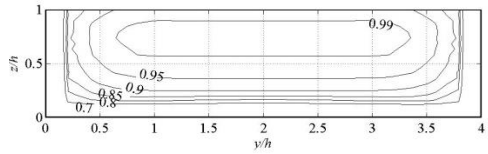


(a) Experimental (b) Numerical

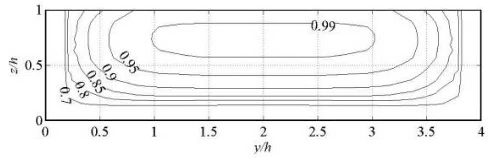
Fig. 2. Comparison of the experimental results of Tominaga et al. (1989) with the numerical results.

4.2. Establishment length

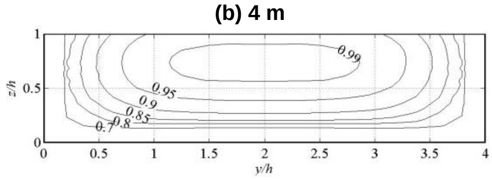
Figure 3 shows the distribution of velocity for four different sections located at various distances from the inlet. In this example, it was observed and concluded that the flow can be treated as fully developed beyond the distance of 6.5 m from the entrance. This means that for these data, the length of boundary layer, i.e., the establishment length is 6.5 m. It represents 65 times the water depth.



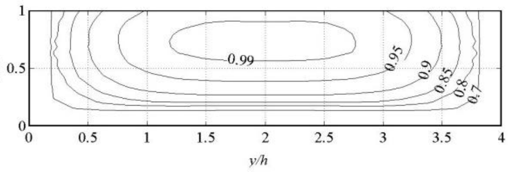
(a) 2 m



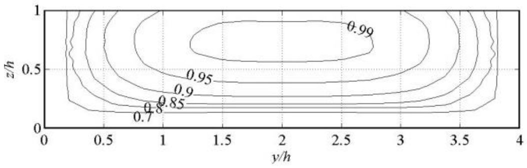
(b) 4 m



(c) 6 m



(d) 6.5 m



(e) 7 m

Fig. 3. Distribution of velocity (U/Umax) at various sections near the entrance of the channel.

Use of a structured mesh provides an identical positioning of the mesh nodes from one section to another. For this feature, the velocity distribution at all points of the successive sections were compared. For the estimation of the establishment length, an indicator of variation was proposed. This is based on the comparison of the velocity values at the corresponding points of the mesh grid in the two consecutive sections as follows:

$$I_{ec}(n,n+1)=\frac{\sqrt{\sum_{i,j\ section}\left(U_{i,j}^{n+1}-U_{i,j}^{n}\right)^2}}{U} \quad (8)$$

where, n and n+1 are the numbers of sections and the summation is performed for any differences between the values of all points Pij of the two sections. The longitudinal development of the mean flow velocity was investigated by calculating this indicator as a function of the distance from the entrance of the channel. This indicator was found to decrease with increase in the distance. The establishment length was defined as the distance from the entrance of the channel where the variation of this indicator values for the two consecutive sections remained within 0.5%.

4.3. Impact of the relative roughness, Reynolds and Froude numbers on the establishment length

Equation (5) shows that the establishment length may be a function of the roughness of the walls (the same roughness value is used for the side wall and channel bed), Reynolds number, and Froude number. In the following section, the influence of these three variables on the length of establishment will be presented.

4.3.1. Influence of the roughness

The changes in roughness of wall produce considerably higher secondary currents than the smooth wall (Demuren and Rodi 1984). Figure 4 shows the distribution of velocity at different distances from the lateral wall for smooth as well as rough walls. The value of y = 0.20 m presents the centerline of cross section. The surface velocity defect (i.e., the difference between the maximum velocity and the surface velocity) near the wall is larger in case of the rough wall compared with that for the smooth wall. Near the rough wall, higher turbulent stresses are generated in contrast to the region over the smooth wall. Therefore, fairly strong gradients of velocity and stresses exist. The flow in the boundary layer is, probably, affected by the development of the turbulent structure produced by the roughness elements.

In general, the roughness slowed the velocity gradient and slightly changed the velocity profiles close to the bed and wall. As the roughness increases, the friction at the wall surface also increases. This increase affects the boundary layer, mainly in the viscous sub-layer and near the walls. The components of the velocity and Reynolds stresses in the outer region (z/h≥0.2) should be affected very little by the roughness (Hinze 1975). Figure 5 indicates the influence of increasing roughness on the velocity field in the developed flow. The effect of roughness on the velocity field remains confined to the region near the wall. However, the average velocity in the outer layer is affected only marginally.

Variation of the velocity gradient with the variation of roughness showed to be more significant near the wall and free water surface. It might be attributed to the rate of dissipation of turbulent kinetic energy.

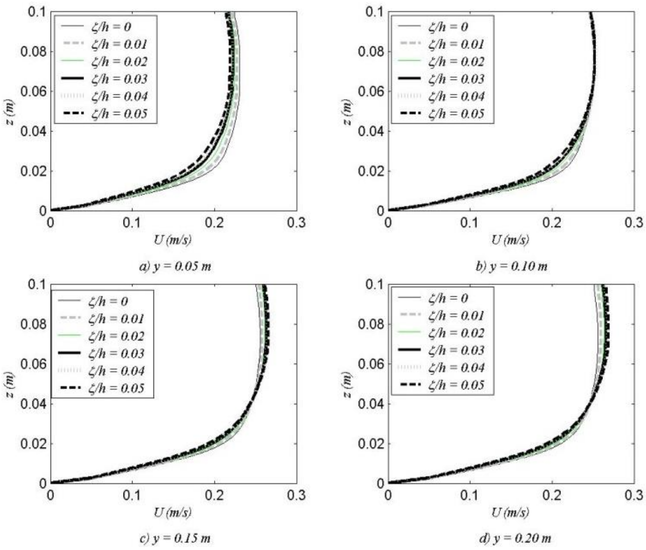
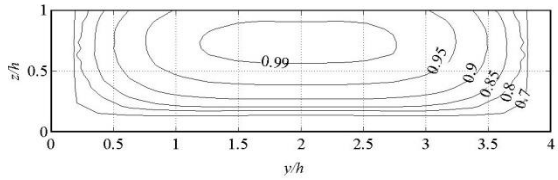
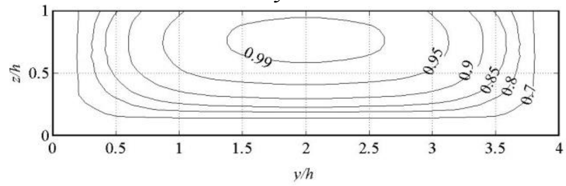


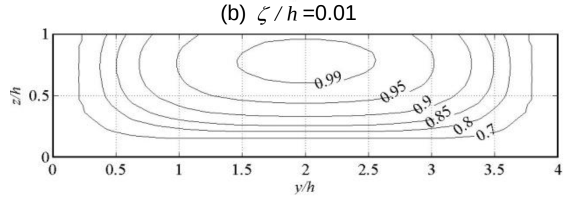
Fig. 4. Variation of velocity at different distances from the lateral wall.



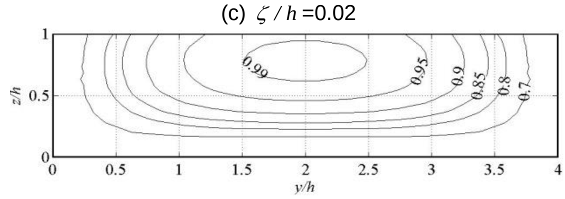
(a) ζ/h =0



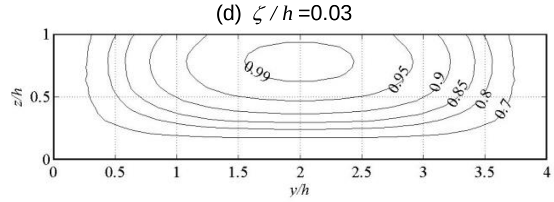
(b) ζ/h =0.01



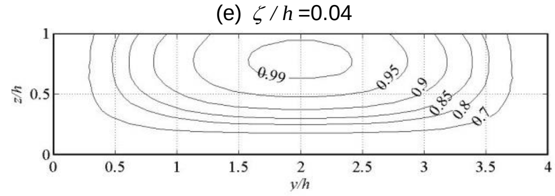
(c) ζ/h =0.02



(d) ζ/h =0.03



(e) ζ/h =0.04



(f) ζ/h =0.05

Fig. 5. Velocity fields (U/U_{max}) for different roughness in the developed flow (x = 6.5 m).

Please cite this article as: H. Bonakdari, G. Lipeme-Kouyi, G. L. Asawa , Developing turbulent flows in rectangular channels: A parametric study, Journal of Applied Research in Water and Wastewater, 1(2), 2014, 53-58.

4.3.2. Influence of the Froude number

As per Eq. 5, the establishment length is likely to be affected by the Froude number. Therefore, the effect of the Froude number is investigated keeping the Reynolds number constant (Eq.5) for the various roughness values. The results indicated that the Froude number has negligible influence on the establishment length. This result is in conformity with the findings of the investigations carried out by Ranga Raju et al. (2000).

4.3.3. Influence of the Reynolds number

Figure 6 presents the dimensionless establishment length as a function of the Reynolds number for different wall roughnesses and for a constant Froude number (= 0.19). As can be seen from Fig. 6, the establishment length increases with a decrease in roughnesses for a specified Reynolds number. Figure 6 also shows that an increase in the Reynolds number causes a decrease in the establishment length.

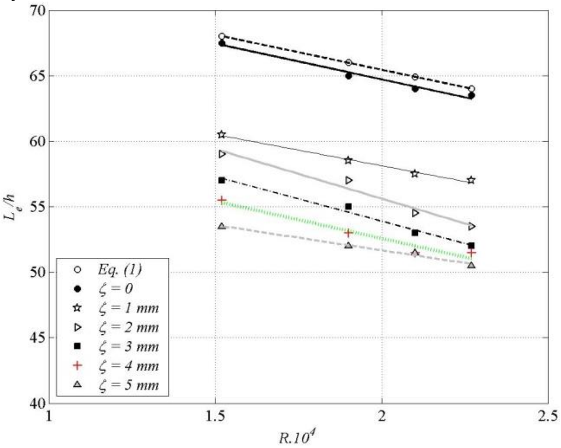


Fig. 6. Variation of Le/h with the Reynolds number for different values of relative roughness (F constant).

Present results, Fig. 6, are almost similar to those given by Kirkgoz and Ardichoglu (1997), Eq.(1), for smooth channels. In the rough conditions, however, one may accept a linear relationship between the dimensionless length of the flow developing zone and the Reynolds number which is as follows:

$$\frac{Le}{h} = aR/F+b \qquad (9)$$

Table 2 shows the evaluated values for the parameters determining the establishment length.

Table 2. Values for the parameters a and b for Eq.(9).

Relative roughness	a×10^{-5}	b
ζ / h =0	-4.8	67.7
ζ / h =0.01	-7.5	70.7
ζ / h =0.02	-6.8	67.5
ζ / h =0.03	-3.8	60
ζ / h =0.04	-5.7	63.9
ζ / h =0.05	-5.5	75.6

5. Conclusions

The entrance length for the fully developed turbulent flow in an open channel was investigated numerically. The establishment length was determined by comparing the modelled velocity fields at successive sections that was substantially different in the establishing zone when compared with the corresponding threshold velocity field of the developed flow zone. The results obtained in the study proved that the effect of the Froude number on the establishment length is negligible. The establishment length was found to decrease with increase in the channel roughness. The dimensionless establishment length showed a linear relationship with the Reynolds number and can be computed using Eq.(9) for which a and b are dependent on the relative roughness, Table 2.

Reference

Akoz M.S., Kirkgoz M.S., Oner A.A., Experimental and numerical modeling of a sluice gate flow. Journal of Hydraulic Research 47(2) (2009) 167-176.

Bonakdari H., Modélisation des écoulements en collecteur d’assainissement – Application à la conception de points de mesures. Thesis (PhD), University of Caen, France, 2006.

Bonakdari H., Zinatizadeh A.A.L., Determination of Doppler flow meters position in sewers using computational fluid dynamics. Flow Measurement and Instrumentation 22 (2011) 225-234.

Bonakdari H., Effects of 3D structured grids on computational modeling of flow. LAP LAMBERT Academic Publishing, Saarbrücken, Germany, 2012.

Cebeci T., Analysis of turbulent flows. ELSEVIER Ltd, Oxford, U.K. 2004.

Cardoso A.H., Graf W.H., Gust G., Uniform flow in a smooth open channel. Journal of Hydraulic Research 27(5) (1989) 603-616.

Dean R., Reynolds number dependence of skin friction and other bulk flow variables in two-dimensional rectangular duct flow. Journal of Fluids Engineering 100 (1978) 215-223.

Demuren A.O., Rodi W., Calculation of turbulence-driven secondary motion in non-circular ducts. Jounal of Fluid Mechanics 140 (1984) 189-222.

Graf W.H., Turbulent characteristics in rough uniform open channel flow. Annual Report, Lausanne, Switzerland, 1991.

Grass A.L., Structural features of turbulent flow over smooth and rough boundaries. Jounal of Fluid Mechanics 50 (1971) 233-255.

Hinze J.O., Turbulence. McGraw-Hill Book Company, Second edition, New York, US, 1975.

Hirt C.W., Nicholas B.D., Volume of fluid (VOF) method for dynamics of free boundaries. Journal of Computational Physics 39 (1981) 201-225.

Kirkgoz S., Ardichoglu M., Velocity profiles of developping and developed open channel flow. Journal of Hydraulic Engineering 115(11) (1997) 1099-1105.

Launder B.E., Spalding D.B., The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering 3(2) (1974) 269-89.

Lien K., Monty J.P., Chong M.S., Ooi A., The entrance length for fully developed turbulent channel flow. 15th Australasian Fluid Mechanics Conference, Ausralia, 2004.

Nezu I., Rodi W., Experimental study on secondary currents in open channel flow. 2lst IAHR congress, Melbourne, Australia, 2 (1985) l9-23.

Nezu I., Nakagawa H., Turbulence in open-channel flows. IAHR-Monograph, A. A. Balkema Publishers, Rotterdam, The Netherlands, 1993.

Nikuradse J., Laws of flow in rough pipes. NACA Technical Notes, No. 1295, Washington D. C., US. 1933.

Perkins H.J., The formation of streamwise vorticity in turbulent duct flow. Journal of Fluid Mechanics 44 (1970) 721-740.

Ranga Raju K.G., Asawa G.L., Mishra H.K., Flow establishment length in rectangular channels and duct. Journal of Hydraulic Engineering 126(7) (2000) 533-539.

Stearns E.P., A reason why the maximum velocity of water flowing in open channels is below the surface. Transactions of the American Society of Civil Engineers 7(1) (1883) 331-338.

Stovin V.R., Grim J.P., Buxton A.P., Tait S.J., Parametric studies on CFD models of sewerage structures. 9th International Conference on Urban Drainage, Portland, US. 2002.

Please cite this article as: H. Bonakdari, G. Lipeme-Kouyi, G. L. Asawa , Developing turbulent flows in rectangular channels: A parametric study, Journal of Applied Research in Water and Wastewater, 1(2), 2014, 53-58.

Tominaga A., Nezu I., Ezaki K., Nakagawa H., Three dimensional turbulent structure in straight open channel flows. Journal of Hydraulic Research 27(1) (1989) 149-173.

Wang J.S., Tullis J.P., Turbulent flow in the entry region of a rough pipe. Journal of Fluids Engineering 96 (1974) 62-68.

Zanoun E.S., Durst F., Nagib H., Evaluating the law of the wall in two-dimensional fully developed turbulent channel flows. Physics of Fluids 15 (2003) 3079–3089.

Please cite this article as: H. Bonakdari, G. Lipeme-Kouyi, G. L. Asawa , Developing turbulent flows in rectangular channels: A parametric study, Journal of Applied Research in Water and Wastewater, 1(2), 2014, 53-58.