HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Conference Paper, Published Version

Shamloo, H.; Azizollahkhani, S.; Pirzadeh, B.

Numerical Modeling of Two-Dimensional Flow Over A Hill In an Open Channel

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Kuratorium für Forschung im Künsteningenieurwesen (KFKI)

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/109834

Vorgeschlagene Zitierweise/Suggested citation:
Shamloo, H.; Azizollahkhani, S.; Pirzadeh, B. (2010): Numerical Modeling of Two-Dimensional Flow Over A Hill In an Open Channel. In: Sundar, V.; Srinivasan, K.; Murali, K.; Sudheer, K.P. (Hg.): ICHE 2010. Proceedings of the 9th International Conference on Hydro-Science & Engineering, August 2-5, 2010, Chennai, India. Chennai: Indian Institute of Technology Madras.

Standardnutzungsbedingungen/Terms of Use:

Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.

Proceedings of ninth International Conference on Hydro-Science and Engineering (ICHE 2010), IIT Madras, Chennai, India.
2 - 5 August 2010

NUMERICAL MODELING OF TWO-DIMENSIONAL FLOW OVER A HILL IN AN OPEN CHANNEL

Shamloo H.[1], S. Azizollahkhani[2] and B. Pirzadeh[3]

Abstract: *This study has been concerned on the turbulent flow over a hill with various submerged ratio i.e. ratio of the flow depth, d, to the body height, h, in an open channel. A two-dimensional numerical mode based on FLUENT software has been used to investigate the wake characteristics downstream of the hills. The RSM and k-ε turbulence models have been employed to simulate the experimental data of Blom (1993). Comparison between numerical and experimental results has shown that the RSM model has greater potential to give accurate predictions for complex flows than the other models. Then a parametric study of flow over a hill has been conducted for various submerged ratios to investigate the effect of this ratio on the velocity field and turbulence intensities. Results indicate that recovery length is a function of the flow depth where increasing d/h causes its decrease. Also, increasing submerged ratio decreases turbulent energy around the hills. The structure of flow in this study has been analyzed using the concept of wall-wake model. Further, modified constants have been suggested for the law of wall downstream of the hills.*

Keywords: *turbulent flow; submerged ratio; recovery length; wall-wake; the law of wall.*

INTRODUCTION

The study of turbulent flow over the bluff-bodies has been an area of interest in many different sectors of engineering such as mechanics, aerodynamics and civil engineering due to its numerous applications. In last decades, a considerable number of studies have been conducted about turbulent flow over surface-mounted obstacles, mostly in wind tunnels and some in open channels. In hydraulic engineering many researches have been conducted on flow around surface piercing cylinders in open channels while our knowledge of flow over other submerged bluff-bodies, such as structures mounted in sea bed and rivers has been remained limited. Moreover, the study of hydraulics of turbulent flow over obstacles at the presence of free surface has been studied even less. The turbulent structure of the back-ward facing step flow in an open channel were measured by Nakagawa and Nezu (1987) by making use of a two-component LDA. In their experiments by changing the water surface elevation, the Reynolds and Froude numbers were varied. The results showed that in a constant Froude number, the reattachment length was

1 Assistant Professor, Department of Civil Engineering, KN Toosi University of Technology, Tehran, Iran, Email: hshamloo@kntu.ac.ir.
2 M.Sc. Student, Department of Civil Engineering, KN Toosi University of Technology, Tehran, Iran, Email: azizollahkhani@sina.kntu.ac.ir.
3 PHD candidate, Department of Civil Engineering, KN Toosi University of Technology, Tehran, Iran, Email: b_pirzadeh@dena.kntu.ac.ir.

decreased with an increase of the Reynolds number and finally was attained a constant value while in a constant Reynolds number; it was increased with an increase of the Froude number. Blom (1993) has conducted an experiment to investigate the turbulent free surface flow over a sill. He measured the velocity profiles, turbulence parameters and Reynolds shear stresses behind the sill. Shamloo et al. (2001) have presented the results of an experimental study on the flow around simple habitat structures. According to the relative depth i.e. the ratio of flow depth to body height, four different regimes of flow have been found. The disturbed flow behind deeply submerged and moderately submerged hemispheres followed the concept of the wall wake. However behind the slightly submerged ones, the mixing was appeared in whole of the flow depth. Finally for a surface piercing body, the Karman vortex stream was found with a strong backward flow behind the body. The experimental results of Sadegh et al. (2008) indicated the same conclusions on the flow around cylinders which were classified based on the relative depth. The main objectives of the present study are to predict the flow behavior past a hill and investigate the influence of submerge ratio on wake characteristics downstream of hills.

EXPERIMENTAL INVESTIGATION

Figure 1 schematically shows measuring stations, S, considered in this study where a turbulent flow passes over a two-dimensional surface mounted hill as it was investigated by Blom (1993). Table 1 shows hill geometry and the hydraulic characteristics of flow where d= depth of flow, U_0= velocity inlet, B= channel width, L= channel length, Q= discharge of flow, h= hill height, Fr= Froude number, Re_d= flow Reynolds number and Re_h= Reynolds number according to the hill height.

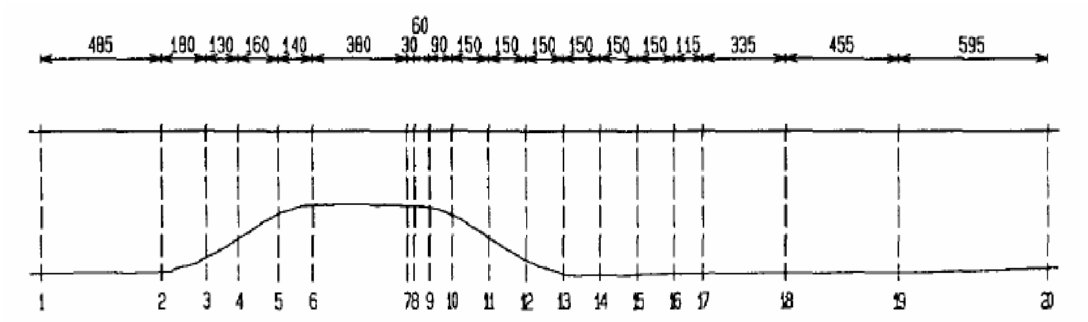


Fig. 1. Schematic of hill and location of measuring stations (Blom 1993).

Table 1. Hydraulic characteristics of the flow

Re_h	Re_d	Fr	h	Q	L	B	U_0	d
17600	35200	0.17	0.08m	17.3L/s	22.5m	0.5m	0.22m/s	0.16m

DOMAIN DESCRIPTION

The upstream inlet is placed at sufficient distance from the hill to ensure that the flow becomes fully developed, where in this study it happens to be 12 meters upstream of the hill. A two phase domain containing flow of water in a channel with a region of air at the top is solved using the multiphase flow model option in FLUENT software. To avoid any effects of the top boundary condition on the results, the ratio between the initial depths of air to the initial depth of water is selected to be two (Salaheddin, 2004) so the channel height is set equal to 0.32 meters. To achieve the accurate results of the flow away from the wall and successful prediction of the bottom shear stresses, the size of the cell adjusted to bed is chosen to satisfy the limits of the wall unit distance, Y*, 11.225<Y*<30 (Fluent user manual, 2006) that Y* is defined by the equation 1:

$$Y^* = \frac{\rho . u^* . y_p}{\mu} \tag{1}$$

where ρ= fluid density, u^*= shear velocity, y_p= distance from point p to the wall and μ= dynamic viscosity of the fluid. In the present study three different meshes with 1095x91 dimensions along with three different Y* equal to 13, 27 and 35 have been selected. Numerical experiments showed that the second one leads to the best results. In the region of rapid variation, the grid structure is fine enough but in other regions it is coarser. The grid generator "Gambit2.3.16" was used for grid generating. The commercial model - FLUENT6.3 - is a CFD solver which solves the Navier–Stokes equations to compute the flow pattern. The present simulations were based on two-dimensional, non-orthogonal, structured grids with a non-staggered variable placement. FLUENT's parallel solver has the ability to compute using multiple processors that may be executing on the same computer, or on different computers in a network to decrease the computing time (Fluent user manual, 2006). This capability was used in the present study using Core2 Due CPU computers. Convective terms in the governing equations were discrete using upwind second order implicit methods and PISO algorithm was also applied for coupling velocity and pressure terms.

In the present study, two separate inlets for air and water were specified and the velocity at both inlets were set as same as associate experimental flow profiles. Also two separate outflows for air and water were specified at the outlet. The velocity was set to be zero at the bottom of channel to satisfy no-slip boundary condition. At the top surface above the air, the normal velocity and the normal gradients of all variables are zero; therefore a symmetry boundary condition is defined there.

RESULTS AND DISCUSSIONS

The main objectives of flow investigation around bodies here is the determination of the flow pattern at wake region due to its important applications in hydraulic engineering. To test the accuracy and validity of the RSM and k-ε turbulent models, a case of turbulent flow over a two-dimensional sill was simulated. The numerical results were compared with the experimental results of Blom (1993) and details are presented as following.

Velocity Profiles

Figure 2 shows the profiles of numerical and experimental stream-wise velocity in different stations, S=6,12,16,20, which are normalized by the initial mean velocity. Due to consideration of a sufficient length of domain upstream of the hill, all turbulence models were able to predict the velocity field in front of the hill well.

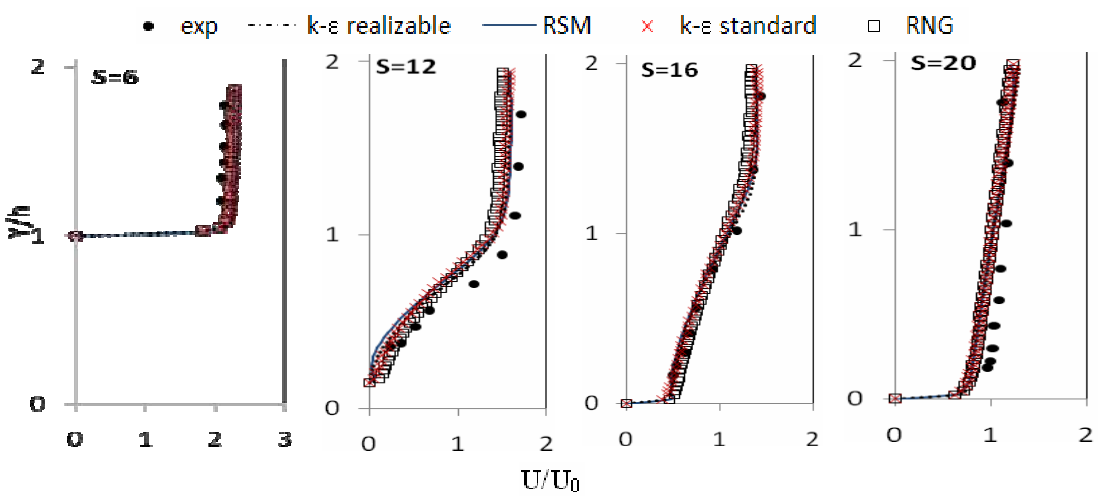


Fig. 2. Comparison of numerical and experimental velocity profiles.

Blom considered the ratio of width to depth of the channel to be three. According to the study by Nezu (2005), to eliminate side walls effects on the middle flow this ratio should be larger than five. So in the experimental study, secondary flow causes the lateral velocity profile to become more uniform. As a result on the top of the hill, the numerical velocities are larger than the measured ones. In this study this deviation for different turbulence models was about 6 percent in the 6th station. Due to the particular geometry of the hill, no separation and circulation region in the flow has been observed and streamlines passed over the hill slightly. In downstream, the comparison of measured and numerical velocity profiles has shown a good agreement. As a result for all four turbulence models, the simulated velocities were underestimated about 10 percent due to the effects of complexity of flow pattern. As the flow progresses further downstream, the disturbed flow behind the hill appears to posses the characteristics of a wake. Finally, behind the hill, at a location far from it (x/h=42), the velocities were recovered to match undisturbed profile gradually.

Turbulent Energy Profiles

In Figure 3 the measured and predicted normalized kinetic energy for various cross-sections are shown. Comparison of turbulent energy profiles for different turbulent models has also shown a general agreement with available experimental data. Results predicted by RSM model apparently fit better with the experimental data than other models at stations around the hill. The reason might be the fact that the k-ε models use the Boussinesq hypothesis in which it assumes μ_t as an isotropic scalar quantity, which is not strictly true.

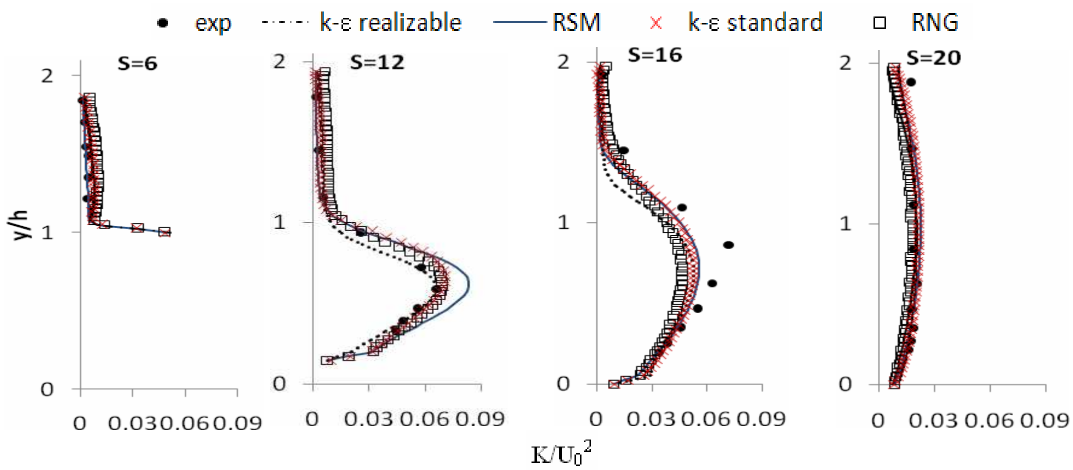


Fig. 3. Comparison of numerical and experimental kinematic energy profiles.

Reynolds Shear Stresses

Comparison between numerical and experimental Reynolds shear stress profiles are presented in Figure 4. Over the hill the agreement between the model results and the experimental data is very good. Near the hill due to the presence of the normal gradient of the velocity, Reynolds shear stresses are increased rapidly which always have negative values. As Figure 4 depicts a maximum value of overestimation of 24% occurs at 14th station. Downstream the hill, decreasing normal gradient of stream-wise velocity decreases the shear stresses which resulted in developing the shear layer along the flow depth gradually.

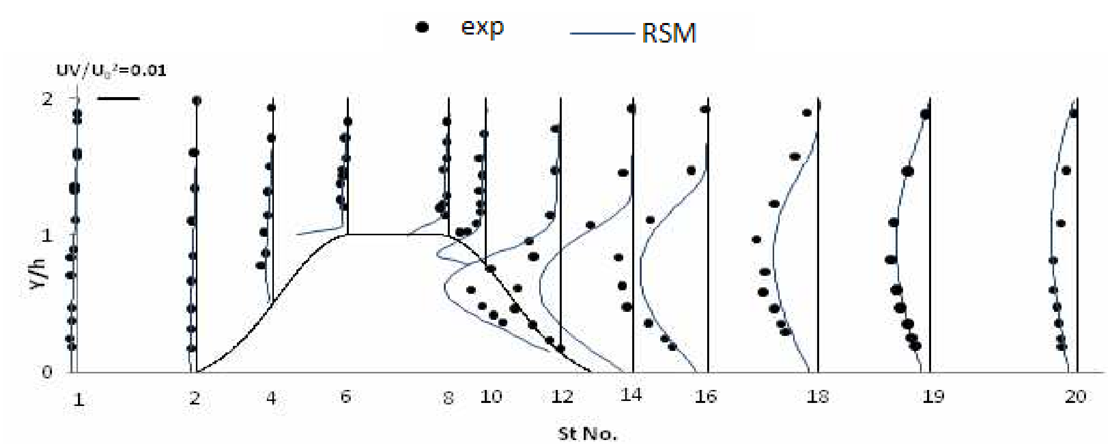


Fig. 4. Comparison of normalized numerical and experimental Reynolds shear stresses profiles.

Effect of Submerged Ratio

In order to investigate effects of submerged ratio on the wake characteristics downstream of the hill, other submerged ratios besides the original ratio in Blom's experiments, 3 and 4, have been also simulated as shown schematically in Figure 5. These experiments were named H25, H35, H45. The first letter is referred to Hill and the second one indicates respectively submerge ratio 2, 3 and 4. The last character indicates a/h i.e. the ratio of top length of hill (a) to its height (h), which is 5 in this study.

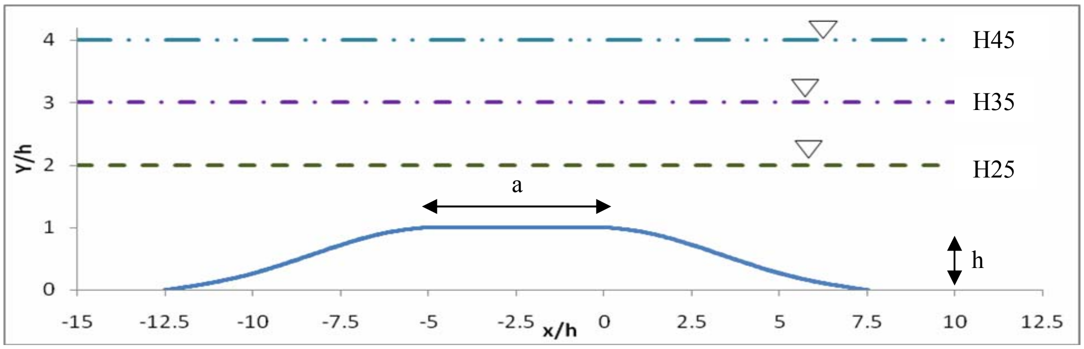


Fig. 5. Schematic of experiments H25, H35, H45.

Recirculation Region behind the Hill

Figure 6 shows the variation of normalized mean bed shear stress downstream of the hill with different submerged ratios. Presence of the special curvature in the hill resulting in low pressure gradients lead to non-separation of the flow. Therefore positive shear stress indicates no flow reversal behind the hills. A comparison between the three cases here shows that with the exception of H25, the normalized value of τ/τ_0=0.2 increases rapidly behind the hill and at x/h=12 it reaches the value of 1. However, in H25, τ/τ_0=0.2 exists for a longer distance, about 7h, then increases to $1.2\tau_0$ at x/h=16.

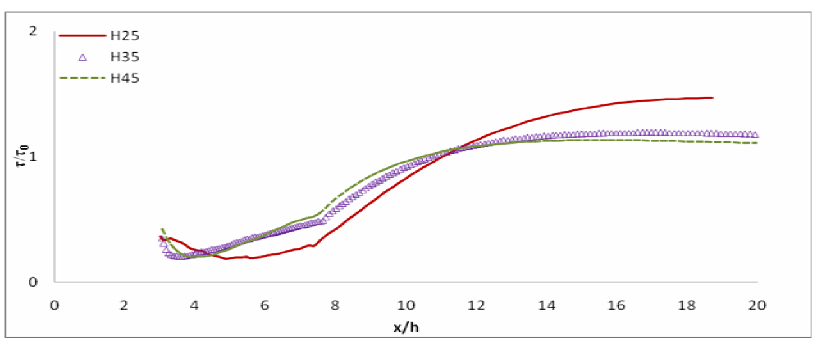


Fig. 6. Downstream Normalized Mean Bed Shear Stress

Velocity Field behind the Hills

Based on Figure 7(a-c), wake behind the hill is changed the velocity field which leads to varied velocity gradients. In contrast, when the relative depth d/h decreases, the velocity gradient increases and develops through the whole depth of the flow. Far from the hill, velocity profiles tend to approach their upstream form. Comparison of the results revealed that the recovery length (Lr) for different cases is inversely related to submerged ratio. The magnitude of Lr was obtained 41h, 34h and 30h for H25, H35 and H45 respectively.

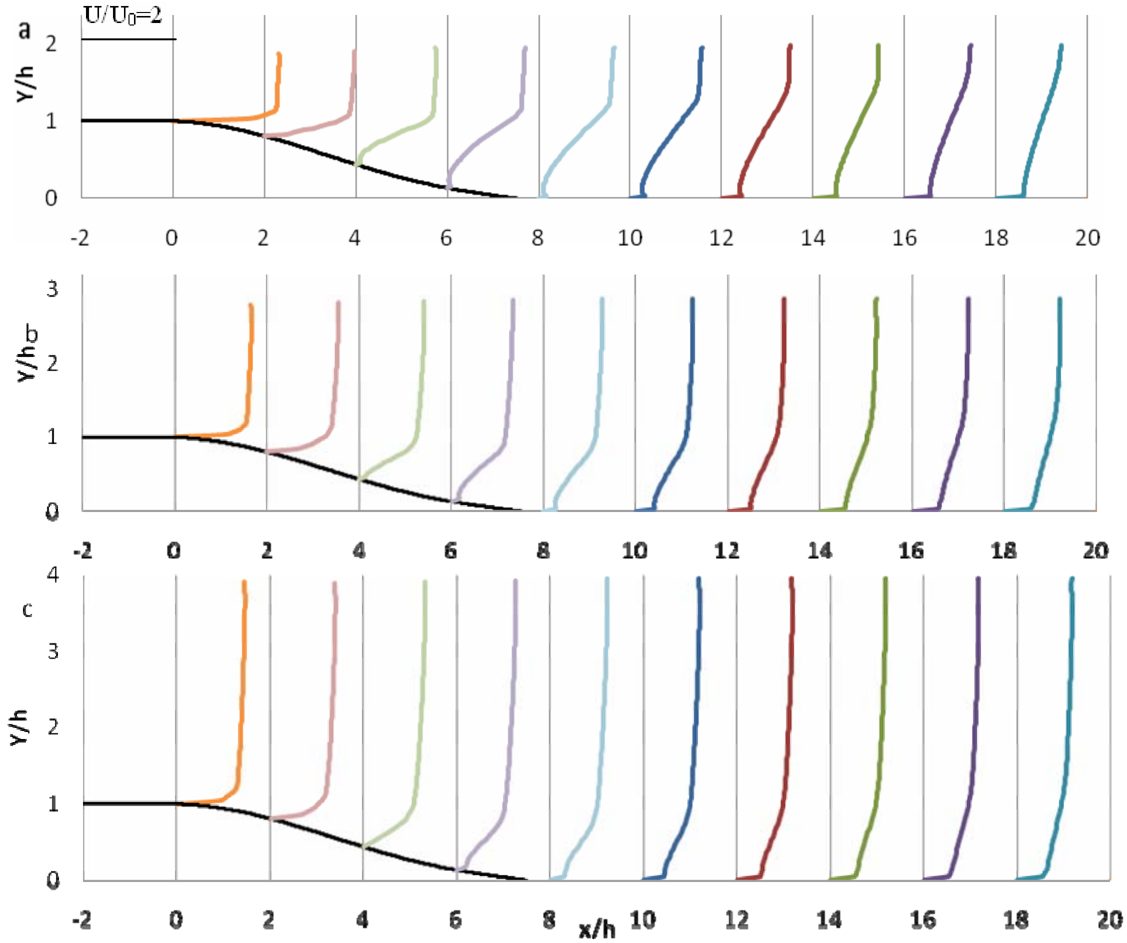
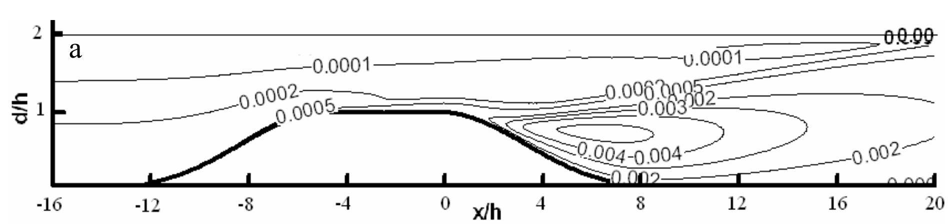


Fig. 7. Downstream Normalized velocity profiles a)H25 b)H35 c)H45

Turbulent Energy Contours

The contours of turbulence energy around the hill are plotted in Figure 8(a-c). This figure indicates that the near-bed turbulence increases considerably when the flow approaches very close to the hill and based on submerged ratio it reaches different maxima as mentioned in table 5. Above the hills, maximum turbulent energy occurs at x/h=-5 as a decrease in submerged ratio leads to increase of turbulent energy. Then there is a significant increase in this quantity at downstream of the hill which is larger for smaller submerged ratios. At 4<x/h<8 the turbulence energy is seen to reach its maxima at different heights from the wall, depending on the case. These heights coincide with the location where the velocity gradient $\partial u/\partial y$ assumes its maxima.



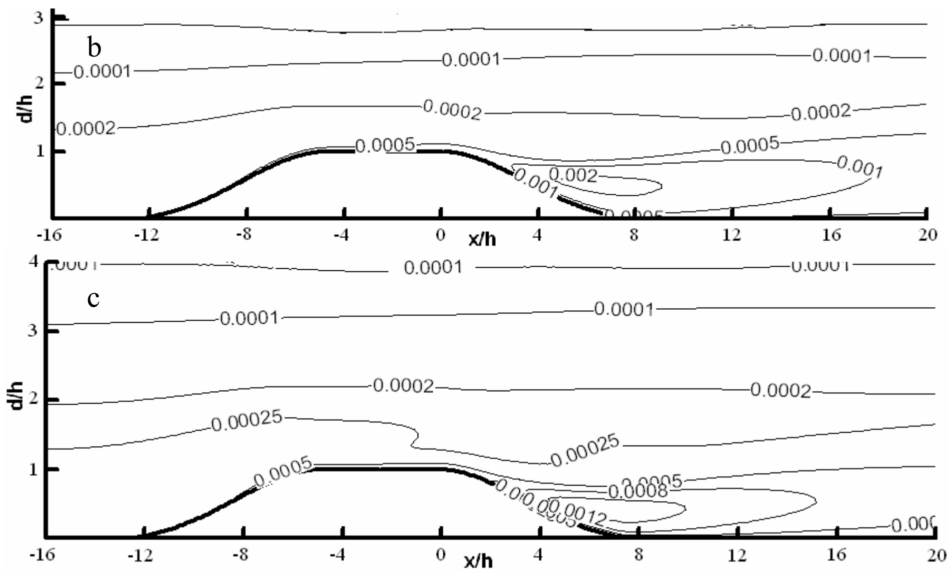


Fig. 8. Kinematic energy contours a)H25 b)H35 c)H45

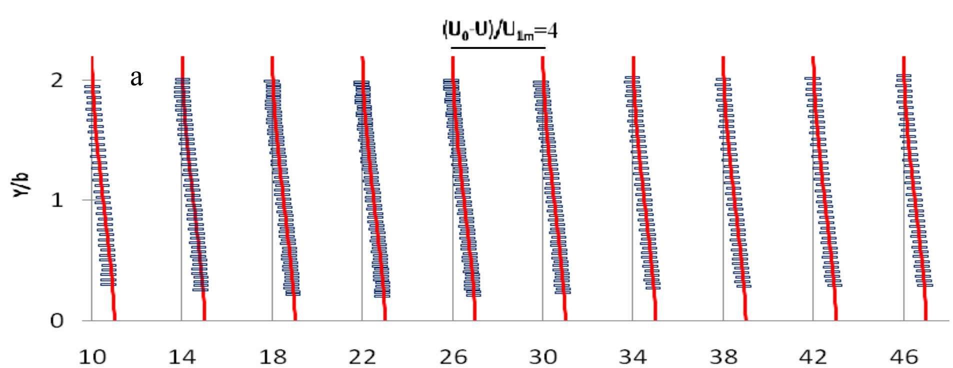
Table 2. Normalized Kinematic Energy

	d/h	Over of hill		Downstream of hill	
		x/h	k/U_0^2	y/h	k/U_0^2
H25	2	-5	5%	0.74	10.9%
H35	3	-5	2.6%	0.5	4.6%
H45	4	-5	2%	0.44	3%

Analysis of Flow in the Wake

The wake characteristics downstream of the hill have been compared with Plane Turbulent Wall-Wake model of Rajaratnam and Rai (1979). Their study presents a two layer model to describe the velocity profiles in the far wake region. In the outer region the velocity profiles are described by the wake equation whereas in the inner region the law of the wall applies. The non-dimensional form of velocity distribution in the outer region is calculated using equation 1 which is shown in Figure 9.

$$\frac{U_0 - u}{u_{1m}} = \left(1 - 0.293.(y/b)^{3/2}\right)^2 \qquad (2)$$



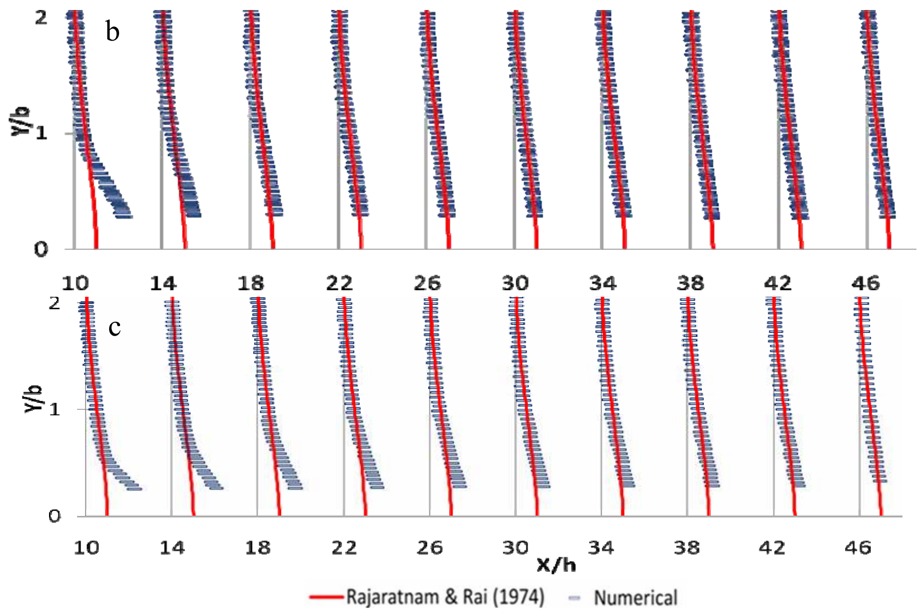
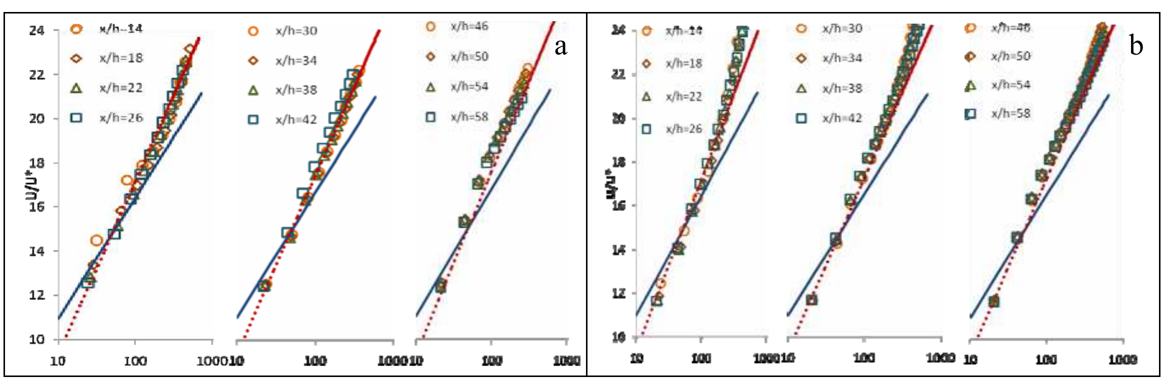


Fig. 9. Comparison of Rajaratnam and Rai's model and numerical results a)H25 b)H35 c)H45

where u_{1m} denotes the maximum value of velocity defect, U_0-u, for a simple wake that it is velocity scale and the length scale is taken as b, equal to y in which u=0.5 u_{1m} (Rajaratnam and Rai, 1979). It can be seen that the velocity profiles attain similarity downstream the hills but the deviation from plane-wake profile starts at about y/b=0.5 close the bed.

In the inner region the velocity profiles were analyzed using the law of the wall which Nezu (2005) and Rajaratnam And Rai (1979) obtained the experimental values, B=3.8-4.9 and A=2.4-5.6. With reference to Figure10, for all cases, profiles are prone to deviation from the law of wall profile perhaps due to possible effects of the hill-wake so we suggested A=7.858 and B=1.48 for flow behind the hills in an open channel. Results showed that the submerged ratio doesn't have any effect on the modified log-law presented in this study.



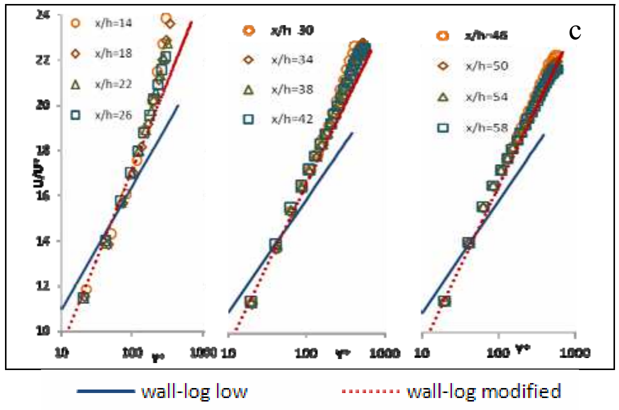


Fig. 10. Comparison of Law of the wall and numerical results a)H25 b)H35 c)H45

CONCLUSION

The turbulent flow past a two-dimensional hill for various submerged ratios has been numerically investigated using FlUENT software. At first, Experimental data (Blom,1993) with submerged ratio 2 was simulated. Compared with k-ε turbulence model, the use of the RSM turbulence model improved predictions of the mean velocities and turbulent energy. The characteristics of the flow around hills depend appreciably on the submerged ratios. Although, there was no recirculation region around the hill due to its particular edged shape, the velocity gradient was varied through the flow depth and the recovery length was inversely related to submerged ratio. Decreasing this ratio causes the increase of the turbulent energy around the hill which reaches its maxima at higher levels from the bottom. These heights coincide with the location where the velocity gradient $\partial u/\partial y$ assumes their maxima. The structure of flow in this study was analyzed using the concept of plane turbulent wall-wake introduced by Rajaratnam and Rai (1979). The results in all cases presented a good agreement in the outer layer but in the inner layer, results deviate from the log-law. This study proposes a modified log-law considering new values for constants in which A=7.858 and B=1.48.

REFERENCES

Blom, P. 1993. Turbulence free-surface flow over a sill. *Ph.D. Thesis. Delft University of Technology*.

Fluent user manual. 2006. Version 6.3.26*. Fluent Inc.*

Nezu, I. and Nakagawa, H. 1993. Turbulence in Open Channel Flow. *Kyoto University, Japan.*

Nezu, I. 2005. Open-Channel Flow Turbulence and Its Research Prospect in the 21st Century*, J. Hydraulic Engineering, ASCE*, 131(4), 229-246.

Rajaratnam, N. and Rai, S.P. 1979. Plane Turbulent Wall Wakes. *J. The Engineering Mechanics Division, ASCE*, 105(EM5), 779-794.

Sadeque, M.A.F., Rajaratnam, N. and Loewen, M.R. 2008. Flow around Cylinders in Open Channels. *J. Engineering Mechanics*, 134(1), 60-71.

Salaheldin, M., Imran, J. and Chaudhry, H. 2004. Numerical Modeling of Three-Dimensional Flow Field around Circular Piers. *J. Hydraulic Engineering*, 130(2), 91-100.

Shamloo, H., Rajaratnam,N. and Katopodis, C. 2001. Hydraulics of simple habitat structures. *J. Hydraulic Research,* 39(4), 351-366.