Comparison of *k-ε* and RNG *k-ε* Turbulent Models for Estimation of Velocity Profiles along the Hydraulic Jump on Corrugated Beds

M. Kaheh
PhD Student, Shahid Chamran University, Ahwaz, Iran

S. M. Kashefipour
Associate professor, Shahid Chamran University, Ahwaz, Iran

A. Dehghani
Assistant Professor, Department of Water Science, University of Agricultural Sciences and Natural Resources, Gorgan, Iran

ABSTRACT: One of the ways to improving the characteristics of hydraulic jump is using corrugated beds in hydraulic jump location. Corrugated beds, by causing strong turbulence in flow, increased the Reynolds stresses, decreased the second depth of jump and as a result affect the depth velocity profile along the hydraulic jump. In this paper, hydraulic jump on corrugated bed is simulated by applying VOF method in Flow-3D Software and using k-ε and RNG *k-ε* turbulent models. The simulated hydraulic jumps in this research are performed in models with 13 mm height and 68 mm wave length of roughness and in a range of Froude numbers from 5 to 7.5. The results indicated that the proposed model was able to accurately model the depth velocity profile along the hydraulic jump. Flow with high shear stress, estimated the second depth, length of jump and velocity distribution of hydraulic jump on corrugated beds relatively well. In this type of hydraulic jump, bed shear stress is higher than its amount in classical hydraulic jump, thus according to the results of this study, the RNG k-ε turbulent model is more suitable for estimating depth velocity profile along the hydraulic jump on corrugated bed

KEYWORD: Hydraulic Jump, Corrugated Beds, Numerical Simulation, Volume of Fluid, k-ε RNG

1. INTRODUCTION

Hydraulic jump is a rapidly varied flow in which the situation of flow is changed from supercritical to subcritical flow. In hydraulic jump on rectangular horizontal channel, many researches have been conducted to obtain hydraulic jump with improved characters. One of the ways to reduce hydraulic jump length is using corrugated bed. Researches have indicated that the height of sequent depth of hydraulic jump on rough beds was significantly reduced. Rajaratnam *et.al* (1968) concluded that for relative roughness d / y_1 of 0.4, the sequent depth of jump is 80 percent of its value on smooth bed. The corrugated bed (as a rough bed) by causing strong turbulent in flow increases the Reynolds stresses and reduces velocity and sequent depth of jump (Ead and Rajaratnam, 2002). Variations in hydraulic jump depth cause the flow pattern to be changed and as a result the velocity profile will be changed. Izadjoo and Safaei-Bajestan (2007) in a series of hydraulic jump experiments on corrugated roughness beds showed a considerable reduction in sequent depth and length of jump. The effect of element roughness shape and height and wave length of corrugation on hydraulic jump characteristics is considered (Abbaspour *et al*, 2009).

Also, many studies were conducted on numerical investigation of hydraulic jump that all of them were on smooth bed. Gunal and Narayanan (1998) in two dimensional models via *k-ε* turbulence model by finite volume method simulated a submerged hydraulic jump. Sabbagh Yazdi *et al* (2007) in a three dimensional model, evaluated *k-ε* and RNG *k-ε* turbulence model on air entrainment into hydraulic jump.

The purpose of this paper is numerical simulation of hydraulic jump on corrugated bed by applying VOF method via *k-ε* and RNG *k-ε* turbulent models in Flow-3D and comparison of the velocity profile predicted by the model and the previous experimental studies in the literature.

2. MATERIALS AND METHODS

Flow-3D is a suitable hydrodynamic model for simulation problem with complex geometry on wide limits of fluid flows in open channel hydraulic. This model employs the VOF method for solving dominant equations on flow at orthogonal mesh gridding. In this model one can simulate the turbulent flows using five turbulent models including, Prantel mixing length, one equation, *k-ε* , RNG *k-ε* and large eddy simulation models equations.

Computational fluid dynamic (CFD) is a method for simulating of flow processes in which the well known flow Navier-Stoks equations and the mass continuity equation are descritized and solve for each computational cell (Tannehill *et al*, 1997). General form of mass continuity equation is:

$$V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + R\frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) + \xi \frac{\rho u A_x}{x} = R_{DIF} + R_{SOR} \tag{2}$$

where, V_F is the fractional volume open to flow, the coefficient R depends on the choice of coordinate system in the following way. Terms of R_{DIF} and R_{SOR} in the right hand side in Equation 2 are related to diffusion turbulent and mass source term, respectively, and are defined as:

$$RDIF = \frac{\partial}{\partial x}\left(\upsilon_\rho A_x \frac{\partial \rho}{\partial x}\right) + R\frac{\partial}{\partial y}\left(\upsilon_\rho A_y \frac{\partial \rho}{\partial y}\right) + \frac{\partial}{\partial z}\left(\upsilon_\rho A_z \frac{\partial \rho}{\partial z}\right) + \xi \frac{\rho \upsilon_\rho A_x}{x} \tag{3}$$

$$\frac{V_F}{\rho c^{۲}}\frac{\partial P}{\partial t} + \frac{\partial u A_x}{\partial x} + R\frac{\partial v A_y}{\partial y} + \frac{\partial w A_z}{\partial z} + \xi \frac{u A_x}{x} = \frac{R_{SOR}}{\rho} \tag{4}$$

Three dimensional forms of the momentum equations can be explained in three dimensions as:

$$\begin{aligned}
&\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z}\right\} - \zeta \frac{A_y v^2}{x V_F} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x - b_x - \frac{R_{SOR}}{\rho V_F}(u - u_w - \delta u_s) \\
&\frac{\partial v}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z}\right\} + \zeta \frac{A_y u v}{x V_F} = -\frac{1}{\rho}\left(R\frac{\partial P}{\partial y}\right) + G_y + f_y - b_y - \frac{R_{SOR}}{\rho V_F}(v - v_w - \delta v_s) \\
&\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z - b_z - \frac{R_{SOR}}{\rho V_F}(w - w_w - \delta w_s)
\end{aligned} \tag{5}$$

where, G, f and b are respectively body acceleration, viscose acceleration and flow losses in porous media or across porous baffle plate.

Two more sophisticated and more widely used-Turbulence models are k-ε, RNG k-ε models. These models were implemented in numerical simulations in this research. The consists of two transport equations for the turbulent kinetic energy k_T and its dissipation ε_T are:

$$\frac{\partial k_T}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial k_T}{\partial x}+vA_y\frac{\partial k_T}{\partial y}+wA_z\frac{\partial k_T}{\partial z}\right\}=P_T+G_T+Diff_T-\varepsilon_T \tag{6}$$

$$\frac{\partial \varepsilon_T}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial \varepsilon_T}{\partial x}+vA_y\frac{\partial \varepsilon_T}{\partial y}+wA_z\frac{\partial \varepsilon_T}{\partial z}\right\}=\frac{CDIS1}{k_T}(PT+CDIS3.G)+Diff_l-CDIS2\frac{\varepsilon_T^2}{k_T} \tag{7}$$

In transport equations for the turbulent kinetic energy (Equation 6) P_T, G_T and $Diff_T$ are transport equations for the turbulent kinetic energy, the buoyancy production and the diffusion term, respectively. The differences of these models are in values of *CDIS1, CDIS2*, *CDIS3* in Equations 6 and 7 (Flow Science, 2009).

Numerical simulations of hydraulic jump on corrugated beds are performed based on a part of the conducted experimental studies by Ead and Rajaratnam (2002). In first, by using AutoCAD software a corrugated plate was designed with the length t and the wave length corrugation s of 13 and 68 mm, respectively. Then the supercritical flows were formed by designing of a gate in upstream of the plate. For preventing of instability and submergence hydraulic jump, a broad crested weir was setup in the end of plate (Figure 1).

Numerical simulations of hydraulic jump on corrugated bed in a range of Froude numbers from 5 to 8 were setup and the model results were compared with the corresponding values collected from the previous existing experimental studies. In computational fluid dynamics, the proper mesh building for obtaining precise results is very important, hence, size of meshes in depth direction were built in three parts (see Figure 1b).

The boundary conditions in upstream, downstream, top and bottom of gridding region were selected as specific velocity, outflow, symmetry and no-slip wall conditions, respectively (see Figure 2).

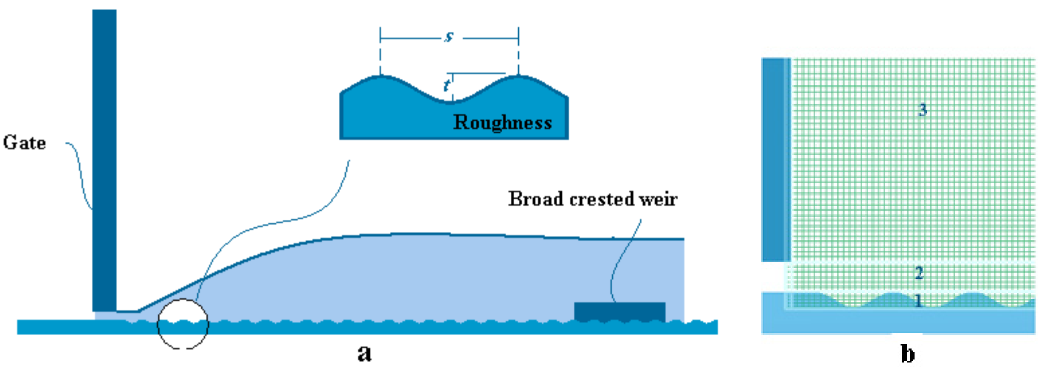

Figure 1. Design of corrugated plate and different parts of gridding

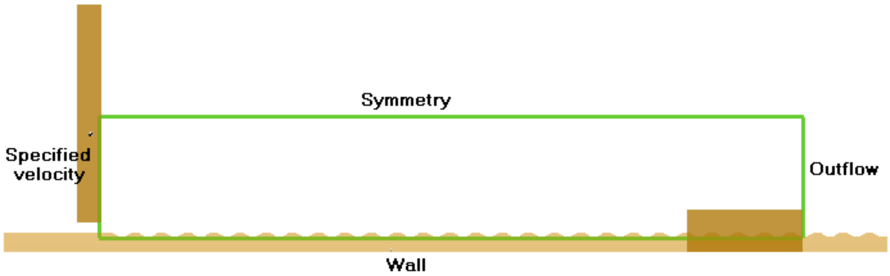

Figure 2. Applied boundary conditions in four sides of gridding region

For accurate modeling, two parameters as maximum aspect ratio (MAR) and maximum adjacent cell size ratio for computational gridding should be close to 1 and not to be greater than 3 and 1.3, respectively (Flow science, 2009).

Further, for computational error convergence and having of steady flow in all numerical simulation, running times were set on a range from 100 to 120 seconds.

3. RESULTS AND DISCUSSION

Having executed the simulated models, the velocity profiles of jump were derived and then compared with the corresponding measured values reported by Ead and Rajaratnam (2002).

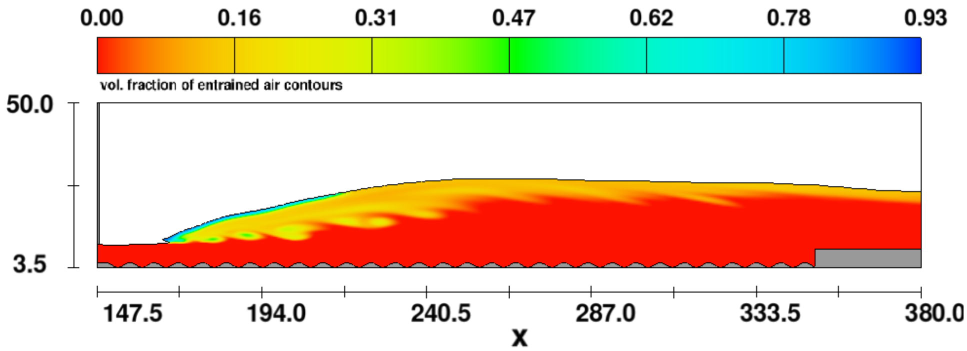

Figure 3 Typical of simulated hydraulic jump on corrugated bed using RNG *k-ε* Model ($q=0.14$ m^2/s and $y_1=5.32$ cm)

Figure 3 depicts an instance of the simulated hydraulic jump on rough bed. Figure 4 shows a typical predicted velocity profiles along a hydraulic jump on corrugated bed obtained from numerical simulation (RNG *k- ε* model). In this profiles the wall-jet type velocity variations, the growth of the boundary layers and the reduction of the maximum velocity with increasing the distance from initial depth of jump are clearly observed. The predicted velocity profiles along a hydraulic jump obtained of two turbulence models applied in this research study are compared together in Figure 5. As can be seen from this figure for *k- ε* model the thickness of wall type velocity is greater than that in RNG *k-ε* model. The dimensionless or normalized forward velocity ($u/u_{\max}$) is also drawn against dimensionless depth (y/b) according to the obtained results from running the Flow3D software using two turbulence considered models for corrugated bed in Figure 6. The corresponding measured values published by Ead and Rajaratnam (2002) regarding this problem are also shown in this figure. Figure 6 shows similarity of the normalized velocity profiles from numerical modeling to that from experimental results, where u_m and b are the maximum velocity and length scale equal to the value of y at which $u = 0.5u_m$ and $\partial u / \sigma y < 0$, respectively. Results show that the RNG k-ε turbulence model has more accuracy as compared to the *k-ε* model to estimate of wall type velocity profiles in near bed at where the shear forces are high.

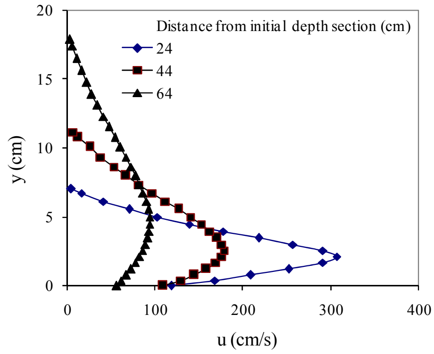

Fig.4. Typical forward velocity profile along a hydraulic jump on corrugated beds (RNG k-ε turbulence model)

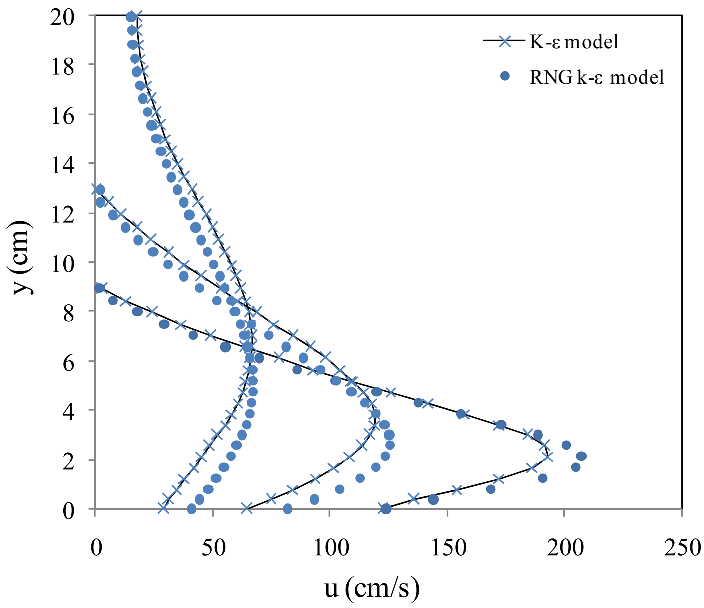

Figure 5. Comparison of simulated forward velocity profiles using *k- ε* and RNG *k- ε* turbulence models

3. CONCLUSIONS

Details are given herein of a numerical study of applying the FLOW3D software for modeling forward profile velocity along a hydraulic jump occurred on a corrugated bed. Two different turbulence models including *k-ε* and RNG *k-ε* models were used in this modeling to determine the capability of these turbulence models for special hydraulic problems such as hydraulic jump on corrugated beds. The experiments carried out by Ead and Rajarrajnam (2002) were simulated in this study. Comparison of the predicted forward velocity profiles along the hydraulic jumps with the corresponding measured profiles showed that both turbulence models were able to accurately simulate this phenomenon, with the RNG *k-ε* model having better agreement with the experimental study. This implies that the later model is better for simulating all problems in which the shear forces are high.

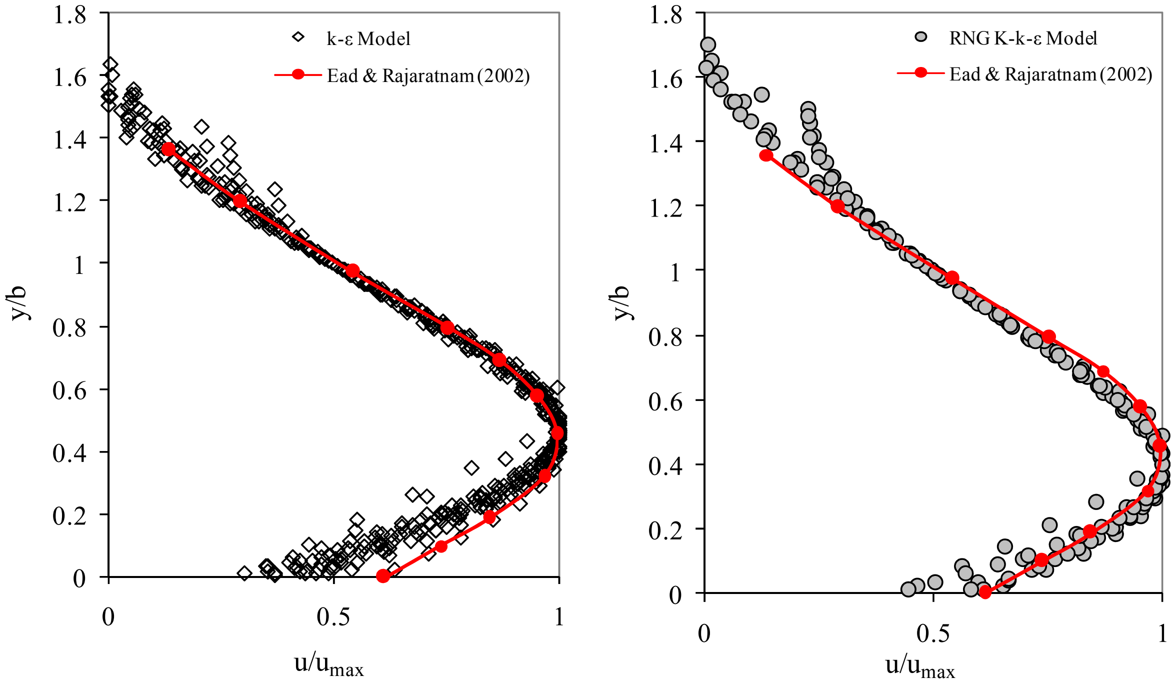

Fig.6. Normalized velocity profiles for all Numerical simulation

4. REFERENCES

Abbaspour. A., Hossein zadeh Dalir. A., Farsadizadeh D. and A. A. Sadraddini, 2009, "Effect of Sinusoidal Corrugated Bed on Hydraulic Jump Characteristics", Journal of Applied Sciences, ISSN 1812-5654.

Ead, S. A., and Rajaratnam, N. 2002, “Hydraulic jumps on corrugated beds.” Journal of Hydraulic Engineering, ASCE, Vol. 128, No. 7, pp. 656-663.

Gunal.M. and Narayanan.R., (1998). "K-ε turbulence modeling of submerged hydraulic jump using boundary-fitted coordinates", Proceedings of the Institution of Civil Engineers. Water, Maritime and Energy, Vol.130, pp.104-114.

Flow Science Inc, 2009, "Flow3D user's manual", SanataFe, N.M.

Izadjoo, F., and Shafai-Bajestan, M. 2007, "Corrugated Bed Hydraulic Jump Stilling Basin", Journal of Applied Sciences, Vol.7 (8), pp. 1164-1169.

Rajaratnam, N., 1968, "Hydraulic Jump on rough beds", Trans. Eng. Inst., Canada, Vol.11(A-2), pp.1-8.

Sabbagh-Yazdi, S. R., Rostami, F. and Nikos E. Mastorakis, 2007, "Turbulent modeling effects on finite volume solution of three dimensional aerated hydraulic jumps using volume of fluid", Proceedings of the 12th WSEAS International Conference on Applied Mathematics.

Tannehill, J.C. , D.A. Anderson, and R.H. Pletcher, 1997, “Computational Fluid Mechanics and Heat Transfer”, 2nd Ed., Taylor & Francis, USA, 792pp