



NUMERICAL MODELLING OF TURBULENT FLOW IN OPEN CHANNEL EXPANSION

Dr. Shaymaa Abdul Muttaleb Alhashimi

Assist Prof., Civil Engineering Department, Mustansiriayah University, Baghdad, Iraq.

Abstract: Open channel expansion is a transition that connects a relatively narrow upstream channel-section to a wide downstream channel-section. Such a transition is an important component of many hydraulic structures. Due to an increase in cross-sectional area, channel expansions cause flowing water to decelerate. Under steady flow conditions, flow deceleration will lead to an increase in water pressure that in turn triggers flow separation and creates turbulent eddy motions. These turbulent eddy motions can exist over a long distance downstream of the transition. They cause undesirable energy losses and sidewall erosion. Computational fluid dynamics (CFD) is used to predict the flow characteristics of open channel expansion. Due to their lower time demand and lower cost, these numerical methods are preferred to experimental methods after they are properly validated. In the present study, the CFD solution is validated by experimental results. A limited number of CFD simulations were completed using the commercial Software FLUENT. In particular, mean velocity distributions for the rectangular open channel transitions were used for model validation. The two-dimensional Reynolds-Averaged Navier-Stokes (RANS) equations and the two equations RNG $k-\epsilon$ models were used. The validation of the model using test data was reasonable. This paper presents the results of experimental investigations on subcritical flow through gradual expansion in rectangular rigid-bed channels employed by CFD for different value of flow. The velocity distributions of flow through the transition models are made, thus, the efficiencies of the transitions evolved by different value of discharge are evaluated.

Keywords: *Open Channel Expansion, Turbulent Flow, CFD Modeling, FLUENT Software, Flow Velocity*

النمذجة العددية للجريان المضطرب في قناة مفتوحة متوسعة المقطع

الخلاصة: التوسع في القناة المفتوحة هو الجزء الانتقالي الذي يربط مدخل القناة ذات المقطع الضيق نسبياً مع الجزء المتوسع في مخرج القناة. هذا الجزء الانتقالي من القناة هو عنصر هام في العديد من المنشآت الهيدروليكية. لذلك زيادة مساحة المقطع العرضي للقناة تسبب تباطؤ في حركة جريان المياه. في ظل ظروف الجريان المستقر، تباطؤ الجريان يؤدي إلى زيادة في ضغط المياه وهذا بدوره يؤدي فصل التدفق ويخلق حركات الدوامات المضطربة. يمكن أن تمتد هذه الدوامات المضطربة على مسافة طويلة في نهاية المقطع الانتقالي وتسبب خسائر الطاقة غير المرغوب فيها وكذلك تآكل في جدار القناة. لتنبؤ خصائص الجريان في المقطع المتوسع لقناة مفتوحة تم استخدام موديل عددي (CFD). يفضل تطبيق هذه الطرق العددية بدلاً عن الطرق المختبرية بسبب قلة الوقت اللازم لانجازه وانخفاض التكلفة، وذلك بعد أن يتم التحقق من صحة نتائجه بشكل صحيح. في هذه الدراسة، تم التحقق من صحة الحل لموديل (CFD) من النتائج المختبرية. تم استخدام معادلة رينولدز-نافير-ستوكس (RANS) ثنائية الأبعاد ونموذج الاضطراب (RNG) وكان التحقق من صحة النموذج باستخدام بيانات الاختبار معقول. قدمت هذه الدراسة قدمت تطبيق الموديل العددي (CFD) على نموذج مختبري لجريان دون الحرجة خلال مقطع التوسع التدريجي في قناة مستطيلة لقيم جريان مختلفة. وتم تقييم الكفاءة للفتوات المتوسعة من خلال تخمين توزيع سرعة الجريان من نتائج الموديل لقيم جريان مختلفه.

1. Introduction

Open channel expansions for subcritical flow are encountered in the design of hydraulic structures such as aqueducts, siphons, barrages, and so on. In these structures the flow tends to separate while subjected to the positive pressure gradient associated with flow deceleration, thus resulting in a considerable loss of energy. Also, the flow tends to separate from its diverging sidewalls and create turbulent eddies, if the angle of divergence exceeds a threshold value. This phenomenon can cause undesirable flow energy losses and erosion to the sidewalls locally and even further downstream.

The issue of flow energy losses in channel expansions is relevant and important in many hydraulics engineering applications Najafi [1]. With a decreasing flow depth and increasing Froude number, the dimensions of the flow separation zones increase, as do the energy losses along the transition. Abbott and Kline [2] observed asymmetric flow patterns on an expansion transition in their experimental investigations. They also found that the Reynolds numbers and turbulence intensities have no effect on flow pattern. Nashta and Grade [3] present the results of analytical and experimental investigations of channels with a sudden expansion for subcritical flow. Swamee and Basak [4] discuss a method for the design of expansions that connect a rectangular channel section with a trapezoidal channel section for subcritical flow. They suggest that flow separation in the expansion and the associated energy losses were considerably reduced. Manica and Bortoli [5] consider laminar flow with a low Reynolds number in a symmetric sudden expansion, numerically. Their results show that below a certain critical Reynolds number (about 50) the flow pattern is symmetric around the channel central line. Hsu, Su, and Chang [6] suggest a method for estimating the optimal length of the contraction transition under supercritical flow using the ratio of downstream to upstream channel width. They conclude that as the ratio of constriction increases the optimal length of the contraction transition decreases.

Alauddin and Basak [7] examined the flow hydraulic in an abrupt expansion, experimentally. The purpose of their study was to design an expanding transition with minimum flow separation and hence small energy head losses. Velocity distributions of flow through the sudden as well as gradual expansion models were made, thus, a transition profile for the expansion of flow with minimum separation was evolved by streamlining the boundary shape of the transition. Haque [8] investigated flow velocity profiles, turbulence kinetic energy and turbulence intensity in expansive transitions, along with the effect of the hump location on bed shear stress. Basak and Alauddin [9] studied the effect of different upstream Froude numbers and inlet discharges on the efficiency of the transition in an expansive transition.

They show that as the upstream Froude number and inlet discharge increase, the efficiency of the transition decreases. As numerical hydraulic models can significantly reduce the costs associated with experimental models, their use has been rapidly expanded in recent decades. Some methods use finite elements (Chau & Jiang, [10]; Wu & Chau, [11]) and others use finite volume in numerical modeling (Epely-Chauvin, De Cesare, & Schwindt,[12]; Liu and Yang,[13]). Howes, Burt, and Sanders [14] studied

flow velocity profiles along the contraction transition for different inlet Froude numbers and inlet positions under subcritical flow, experimentally.

The problem of turbulent flow in an open channel expansion is very complicated, with turbulent eddy motions, flow separation and so on. It is difficult to use the analytical approach to obtain solutions to the problem even under highly simplified conditions. The experimental approach may be taken to tackle the problem, but experiments are very expensive to carry out. In this study, the CFD modelling approach is employed. A numerical simulation was developed using the finite-volume method with RNG $k-\varepsilon$ turbulence model. After calibration, the created secondary currents and effect of the different inlet discharges on the created separation zones at the transition corners were investigated.

2. Numerical Modeling

2.1. Governing Equations

Governing Navier-Stokes equations are employed for developing a two-dimensional model for unsteady flow derived from conservation of mass and momentum equations consisted of velocity and pressure formulation of the incompressible fluid, which is described as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \quad (1)$$

where ρ is the density and v is the velocity of the fluid. The conservation of momentum is similarly described by the equation:

$$\frac{\partial}{\partial t} (\rho v) + \nabla \cdot (\rho v v) = -\nabla p + \nabla \cdot \tau \quad (2)$$

where p is the pressure and τ is the stress tensor. In order to represent the effects of turbulence on the flow, additional transport equations are solved for various turbulence quantities.

In the present work, the geometric reconstruction method of Yeoh and Tu [15] was employed. To model upstream and downstream boundaries, CFD software includes boundary conditions that are specific to the open channel case, at which the upstream and downstream water levels can be specified.

2.2. CFD MODEL

The simulations model used version 6.2 of FLUENT, Fluent Inc. [16] applied on a hydraulic model produced by Najafi [1], as shown in Fig.1. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT. This software uses a control-volume-based technique to convert a general scalar transport equation to an algebraic equation that can be solved numerically. Pressure-

velocity coupling is achieved by using five algorithms: SIMPLE, SIMPLEC, PISO, Coupled and Fractional Step.



Figure 1. Channel expansion and pressure taps through the wall and connected to piezometers.

A transient numerical model was applied owing to the use of the geometric reconstruction surface tracking algorithm. The Re-Normalised Group theory (RNG) $k-\epsilon$ turbulence model of Yakhot and Smith [17] was used with standard wall functions. This is one of a range of turbulence models classed as Reynolds-Averaged Navier-Stokes (RANS) models as defined by Versteeg and Malalasekera [18]. They are time-averaged approximations that are widely used in industrial applications. The RNG $k-\epsilon$ has known advantages when there is strong curvature in the streamlines, as is the case with the decelerating flow through expansion therefore was used. To complete the description of the CFD modelling: the standard pressure discretization scheme was used because of the presence of gravity; second-order discretization schemes were employed for the momentum, turbulence kinetic energy and dissipation equations. The pressure velocity coupling algorithm was applied SIMPLE because it is designed specifically for transient simulations. A time step of 1.0×10^{-4} s was used throughout to keep the simulation stable model. Since the flow fields being modelled take of the order of 10 s to establish, it can be appreciated how many time steps are required to reach steady-state conditions. Figure 2, shows the dimensions of the domain and position of the boundary conditions used in the modelling. At inlet the upstream boundary velocity inlet was used. At the downstream pressure outlet, only the free surface height (or tail water) height was required. All other unmarked boundaries are set as walls. On the walls, the no-slip condition was applied and the walls were assumed to be smooth, since the experimental channel was constructed of PVC and glass.

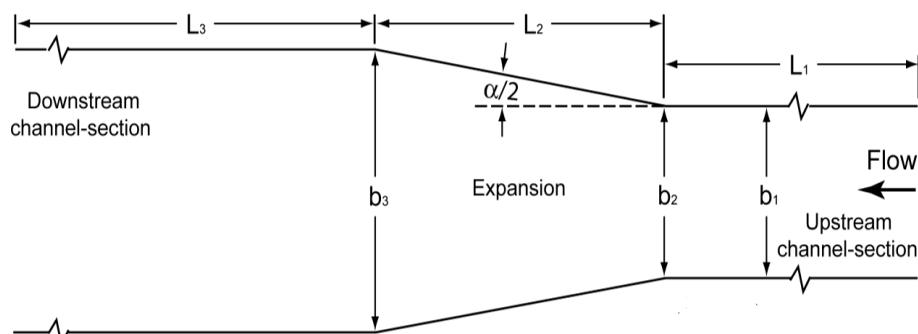


Figure 2. Definition diagram of a channel expansion. Najafi [1].

2.3 Boundary Condition

FLUENT model employed on model presented by Najafi (2011) as shown in Fig 2. The entrance canal is 171.1 mm wide and 1.235 m long and the expanded canal are 289.3 mm wide and 323.3 mm long. There is no bed slope in the canal. The upstream entrance velocities are $u = 0.29$ m/s, $v = 0$ and the flow depth $y = 0.10$ m as the downstream boundary condition is considered. In accordance to Fig. 1, the velocity inlet along with inlet turbulence intensity and hydraulic depth are the upstream boundary condition and pressure outlet is the downstream boundary condition. Boundary condition for bottom and side wall of canal is wall. Due to the turbulent flow, turbulence models were used. SIMPLE algorithm for coupling the velocity field and pressure is used.

At first, it is required to verify the accuracy of meshing, correctness of the choice of turbulence model and ensure that they have no effect on the results. The grid structure must be fine enough, especially near the wall boundaries and the expansion, which is the region of rapid variation. The generated mesh by Gambit is shown in Fig. 3.

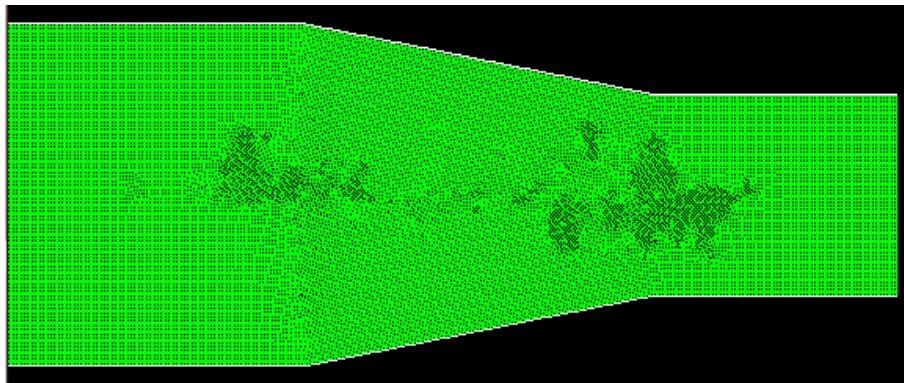


Figure 3. Meshing and its Distribution in the Model of expansion channel

3. Results and discussion

3.1 Validation of Numerical Model

In this research, it is required that the accuracy of the results obtained from numerical study to be verified. The measured and numerical flow is compared by computing the percentage of error from equation (3), as shown in Table 1.

$$Er = \left| \frac{Q_{exp.} - Q_{num.}}{Q_{exp.}} \right| * 100 \quad (3)$$

It can be seen that the results of numerical model for flowrate are greater than those obtained by experiments. The minimum and maximum error values were 3.69% and 5.6% respectively.

Table 1. % of error between experimental and numerical results.

Run No.	$Q_{exp.}(l/s)$	$Q_{num.}(l/s)$	% Er.
1	8.59	8.91	3.72
2	9.47	9.82	3.69
3	10.57	11.01	4.2
4	11.52	12.17	5.6
5	11.81	12.41	5.08

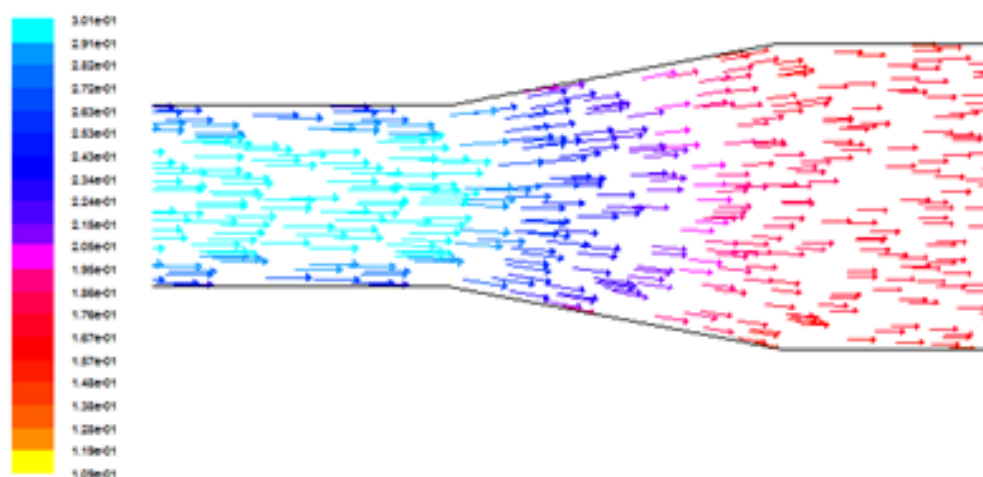
3.2 Simulated flow Pattern in Sudden Expansion

In order to examine more closely the flow pattern in the open channel expansion, fluent numerical model is applied on constructed laboratory flume and a series runs of different value of discharge were performed on it to predict the velocity distribution along the expansion canal. The velocity distribution and contour along channel for different value of flow are shown in Fig. 4 to Fig. 12.

According to Fig. 4, with the exception of $Q = 8.91$ l/s the flow is seemed symmetrical with respect to the centerline of the channel. The maximum velocity line coincides with the center line of the channel for a short length after entry. Thereafter, it shifts to the side to which the main flow is attached.

The reason for the deviation between the maximum velocity line and centerline is related to the asymmetrical circulation and secondary flow that occurred in the cross sections near the wall. The maximum velocity line and centerline do not match, due to the flow separation zones and secondary flow created at transition corners.

For discharges greater than 8.91 l/s, velocities at transition corners are decrease and close to zero, indicating that the flow separation zones are formed there. Also, along the transition, the velocities decrease with approaching the transition outlet.

Figure 4. Velocity vector along the channel for $Q = 8.91$ l/s.

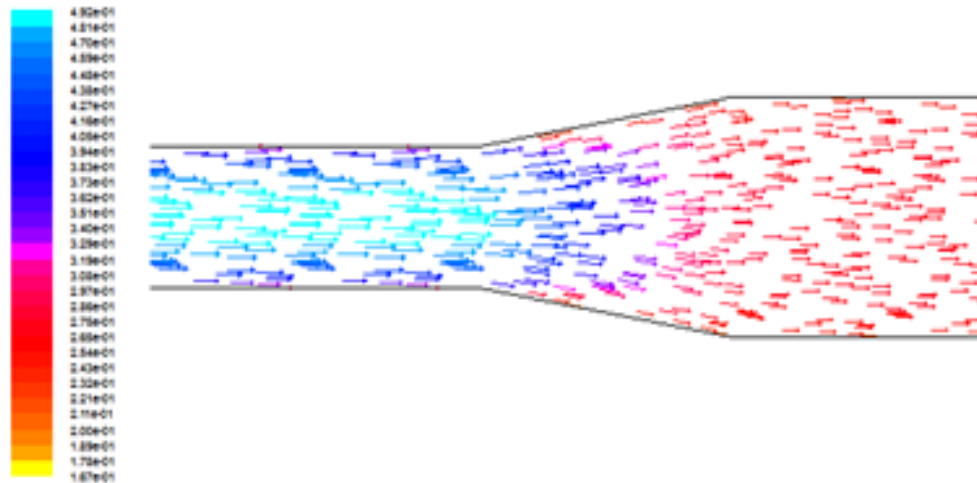


Figure 5. Velocity vector along the channel for $Q = 9.82$ l/s.

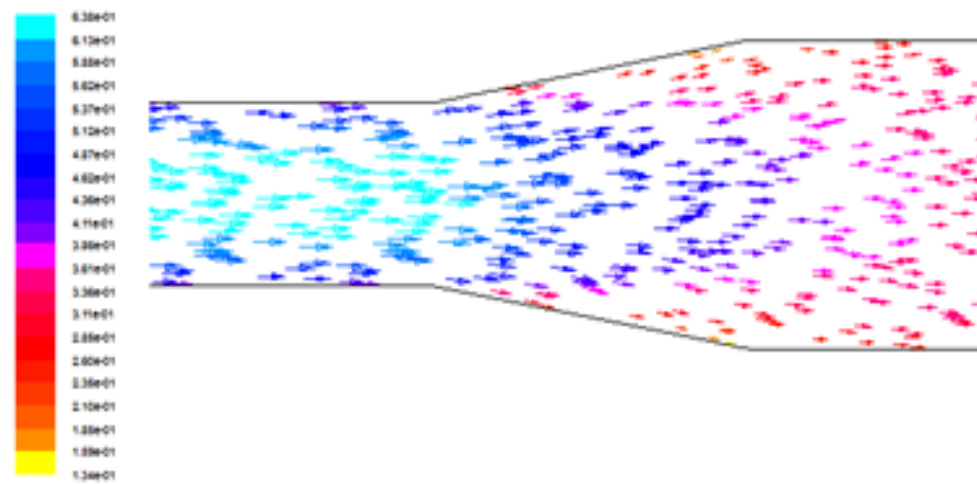


Figure 6. Velocity vector along the channel for $Q = 11.01$ l/s.

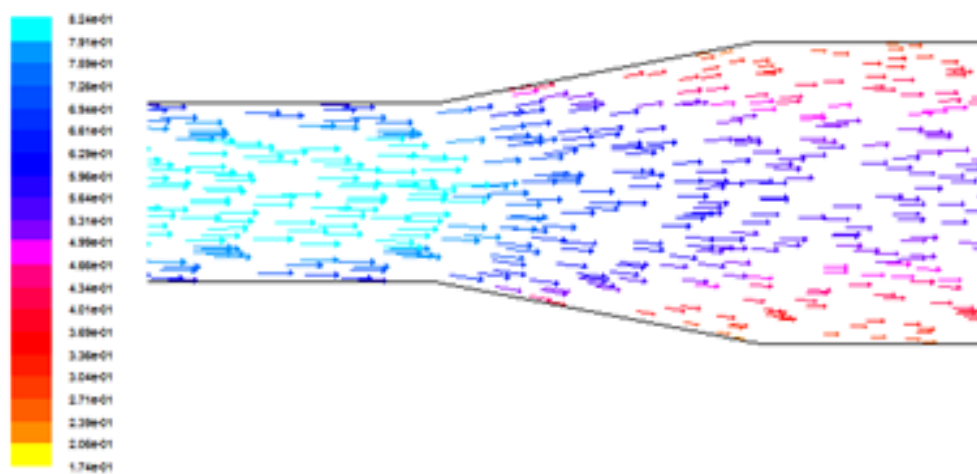


Figure 7. Velocity vector along the channel for $Q = 12.17$ l/s.

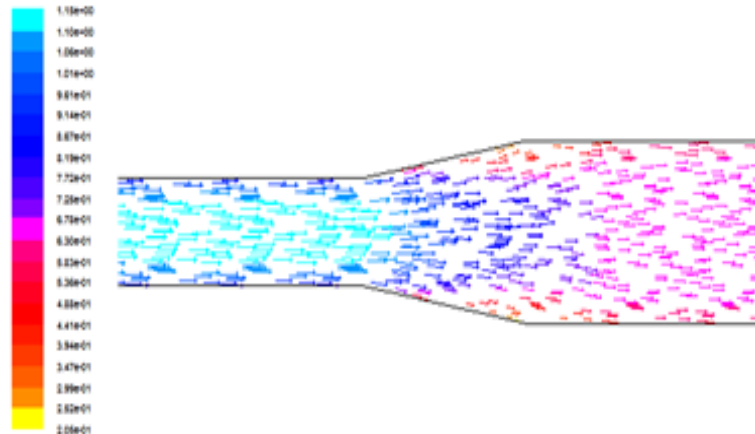


Figure 8. Velocity vector along the channel for $Q = 12.41$ l/s.

As seen in Figure 5 and other as the inlet discharge of the expansion increases, the length and width of the flow separation zone increase as well. Thus it can be concluded that flow separation.

4. Conclusions

This study presents a simulation between the results of experimental models for the subcritical flow through the transition in a rectangular open channel expansion and numerical model. A numerical modelling was performed in two dimensional Reynolds-averaged Navier–Stokes (RANS) equations were solved using a finite-volume method. The flow calculations were employed using RNG $k-\epsilon$ turbulence model. The model results are validated using available experimental data for a limited number of discharges. A good agreement was found between the numerical simulation of the discharge value and the experimental results. From the results of this research can be concluded:-

- 1- CFD model provides predictions of two-dimensional turbulent flow in an open-channel expansion with reasonable level of accuracy.
- 2- Flow pattern in the expansion channel was investigated to be somewhat symmetrical. Symmetric flow pattern is obtained in these conditions for different value of discharge.
- 3- The maximum velocity line coincides with the centerline of the channel for a short length after entry. Thereafter, it shifts to the side to which the main flow is attached.
- 4- The reason for the deviation between the maximum velocity line and centerline is related to the asymmetrical circulation and the secondary flow that occurs in the cross-sections near wall.
- 5- The inlet discharge of expansion has an effect on the flow pattern in the expansion. As the inlet discharge of the expansion increases, the length and width of the flow separation zone increase as well. It was shown for discharges greater than 8.91 l/s, the velocities at transition corners were decrease and close to zero, indicating that flow separation zones are formed here.

5. References

1. Najafi-Nejad-Nasser, A. (2011). *"An experimental investigation of flow energy losses in open-channel expansions"*. Msc Thesis, Civil Engineering, Concordia University.
2. Abbott, D. E., & Kline, S. J. (1962). *"Experimental investigation of subsonic turbulent flow over single and double backward facing steps"*. Journal of Basic Engineering, 84, 317.
3. Nashta, C. F., & Grade, R. J. (1988). *"Subcritical flow in rigid bed open channel expansions"*. Journal of Hydraulic Research, 26(1), 49–65.
4. Swamee, P. K., & Basak, B. C. (1992). *"Design of trapezoidal expansive transitions"*. Journal of Irrigation and Drainage Engineering, 118(1), 61–73.
5. Manica, R., & Bortoli, A. L. (2003). *"Simulation of incompressible non-newtonian flows through channels with sudden expansion using the power-law model"*. TEMA - Tendências Em Matemática Aplicada e Computacional, 4(3).
6. Hsu, M., Su, T., & Chang, T. (2004). *"Optimal channel contraction for supercritical flows"*. Journal of Hydraulic Research, 42(6), 639–644.
7. Alauddin, M., and Basak, B. C. (2006). *"Development of an expansion transition in open channel sub-critical flow"*. Journal of Civil Engineering, 34(2), 91–101.
8. Haque, A. (2008). *"Some characteristics of open channel transition flow"*. Msc Thesis, Civil Engineering, Concordia University.
9. Basak, B. C., and Alauddin, M. (2010). *"Efficiency of an expansive transition in an open channel subcritical flow"*. DUET Journal, Dhaka University of Engineering & Technology, 1(1), 27–32.
10. Chau, K. W., & Jiang, Y. (2001). *"3D numerical model for Pearl river estuary"*. Journal of Hydraulic Engineering, 127(1), 72–82.
11. Wu, C. L., & Chau, K. W. (2006). *"Mathematical model of water quality rehabilitation with rainwater utilization a case study at Hai gang"*. International Journal of Environment and Pollution, 28(3-4), 534–545.
12. Epely-Chauvin, G., De Cesare, G., & Schwindt, S. (2014). *"Numerical modelling of plunge pool scour evolution in non-cohesive sediments"*. Engineering Applications of Computational Fluid Mechanics, 8(4), 477–487.
13. Liu, T., & Yang, J. (2014). *"Three-dimensional computations of water-air flow in a bottom spillway during gate opening"*. Engineering Applications of Computational Fluid Mechanics, 8(1), 104–115.
14. Howes, D. J., Burt, C. M., & Sanders, B. F. (2010). *"Subcritical contraction for improved open-channel flow measurement accuracy with an upward-looking ADV"*. Journal of Irrigation and Drainage Engineering, 136, 617–626.
15. Yeoh G H and Tu J (2010). *"Computational techniques for multiphase flows"*, Elsevier Ltd., USA, pp. 407-500.
16. Fluent Inc, (2006), "User's guide," ver. 6.3.26.
17. Yakhot V and Smith L M (1992), *"The Renormalization Group, the expansion and Derivation of Turbulence Models"*, Journal of Scientific Computing, Vol. 7, pp. 35-61

18 .Versteeg, H. K., & Malalasekera, W. (2007). *"An introduction to computational fluid dynamics"*. 2nded. London: Prentice Hall.