HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Conference Paper, Published Version

Ilkhanipour Zeynali, Rasoul

Two Jets Flow Attack-Induced Turbulence

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

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/110232

Vorgeschlagene Zitierweise/Suggested citation:
Ilkhanipour Zeynali, Rasoul (2008): Two Jets Flow Attack-Induced Turbulence. In: Wang, Sam S. Y. (Hg.): ICHE 2008. Proceedings of the 8th International Conference on Hydro-Science and Engineering, September 9-12, 2008, Nagoya, Japan. Nagoya: Nagoya Hydraulic Research Institute for River Basin Management.

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.

TWO JETS FLOW ATTACK-INDUCED TURBULENCE

Rasoul Ilkhanipour Zeynali

Instructor and researcher, Department of water Engineering, Urmia university
Orumieh, Iran, e-mail: ilkhanipour_bayram@yahoo.com

ABSTRACT

In this paper, the effects of three different angles of two jets flow attack on turbulence introducing are discussed. When two jets flow angle of attack is large and their velocities are so high that they cut together, turbulence and energy dissipation are increased. Turbulence and energy dissipation decrease due to two jets flow opposite attack. Because random collision and scattering of fluid particles introduce more turbulence and energy dissipation, which is not such conditions on opposite attack of two jets flow.

Keywords: collision, attack induced turbulence, Ilkhanipour model, energy dissipation.

INTRODUCTION

Hydraulic jump is formed due to the collision between supercritical and subcritical water flows in the same direction which causes supercritical flow kinetic energy changes, roll up the supercritical flow and eddy formation. Thus, the energy of water will dissipated as a result of the collision between two jet flows. This is the most effective way to shorten a stilling basin without the addition of appurtenances in the basin.

Two multi-particles systems collision change the momentum of the systems. It result an external force on the systems.

$$F=dp/dt \qquad (1)$$

This external force changes resultant force acting on the system. Thus, mean velocity of the system is changed.

$$\sum F=m\,dv/dt \qquad (2)$$

Kinetic energy difference before and after collision is converted to heat.

$$E'_k - E_k = Q' \qquad (3)$$

Energy dissipation increases with increase the angle of collision. Maximum energy dissipation is occurred when the angle of opposite collision is π radians. Moreover, scattering is generated exceed turbulence and conversion of energy to heat (Alonso & Finn, 1967).

Energy dissipation in hydraulic jump results in part from the generation of large scale turbulence and the related conversion of energy to heat (Fiorotto and Rinaldo, 1992).

The velocity, direction of motion and streamline curvature are changed due to collision. Fluid particles scattering and rotation introduce regions with high vortices and shear stresses which cause energy dissipation.

Several studies have shown that there exists an optimal slant angle in terms of minimizing vortex and drag for the rear part of the roof or rear wind-screen (Ahmed, 1983).

Highly turbulent air-water flows skimming down a large-size stepped chute were systematically investigated with 22° slop. Turbulence manipulation was conducted using vanes or longitudinal ribs to enhance interactions between skimming flows and cavity recirculation regions. The results demonstrated the strong influence of vanes on the air-water flow. An increase in flow resistance was observed consistently with maximum flow resistance achieved with vanes placed in a zigzag pattern (Gonzalez and Chanson, 2005).

Turbulent flows contain an organized part, the coherent vortices, and a random part, the incoherent background flow. The separation into coherent and incoherent contributions is done using the wavelet coefficients of vorticity field and the Biot-Savart kernel the coherent and incoherent velocity fields. The evolution of the coherent part is computed using a wavelet basis, adapted at each time step to resolve the regions of strong gradients, while the incoherent part is discarded during the flow evolution, which models turbulent dissipation. The CVS method is similar to LES technique, but it uses nonlinear multiscale band-pass filters, which depend on the instantaneous flow realization, while LES technique uses linear low-pass filters, which do not adapt to the flow evolution (Farge and Schneider, 2001).

The RVM was used to describe the strong dynamics of the flow due to the interaction between an inner jet and an external flow separated by two symmetrical flat plates. The flow was unsteady two dimensional (plane) and incompressible. The study was oriented toward the characterization of the formation and transport of eddy structures in the flow domain. They observed that, in spite of the random nature of turbulence the creation of vortical structures downstream of the obstacle, their mutual interaction just before the separation from the obstacle and their transport by the main flow follows a quasi-coherent logic which can be analyzed by instantaneous observations of the fluid patterns (Mortazavi and Giovannini, 2001).

The model of (Younis et al. 2004) is capable of directly reproducing the effects of streamline curvature on the turbulent scalar fluxes without recourse to ad–hoc modifications.

A complete description of a viscous, turbulent, trailing vortex requires solution to the full Navier-Stokes equations. Analytical solutions to these non-linear sets of equations are not possible, whereas numerical solutions are deterred by formidable computational costs as well as building satisfactory turbulence models. Closed-form solutions for vortex flows can only be obtained by further simplifying the governing equations (Burgers, 1948; Newman, 1959 and Ogawa, 1993).

Several semi-empirical models have been developed and have been applied to the modeling of trailing vortices with generally good results (Vatistas, 1998 and 1991). The strongly self-similar structure of the tip vortices suggest a further simplification based on the assumption of an axisymmetric flow. The vortex length scale dictated by the viscous core radius can then be modeled empirically and independently from the representation of the velocity profile.

A common approach is to represent the induced velocity using a desingularized algebraic profile, with a constant viscous core size or a diffusive core growth with that is based on the Lamb-Oseen model. Algebraic models for the vortex induced velocity profiles are popular in engineering applications because of their simplicity and computational efficiency.

The diffusive growth of a viscous vortex has been modeled using an extension of the classic Lamb-Oseen core growth model with an average apparent or eddy viscosity correction for the effects of self-generated turbulence. The apparent viscosity parameter, as given by Squire's hypothesis, was estimated based on several sets of experimental results documenting the characteristics of trailing tip vortices. A family of algebraic models for the three components of velocity induced by a viscous trailing tip vortex has proposed. The velocity components were determined by solving a simplified form of the incompressible Navier-Stokes equations (Bhagwat and Leishman, 2002).

In the present work, the study is focused on attack-induced turbulence due to collision between a jet flow from an orifice which located on horizontal plane in the toe of the weir and the nappe of water flowing over the weir in Ilkhanipour model for diversion dams (Ilkhanipour, 2007).

The roles played by turbulence in Ilkhanipour model for diversion dams are similar in the atmosphere and oceans which can be classified into two categories: momentum transfer and scalar mixing. In transporting momentum, turbulent motions behave in a manner roughly analogous to molecular viscosity, reducing differences in velocity between different regions of a flow.

Scalar mixing refers to the homogenization of fluid properties such as temperature by random molecular motions. Molecular mixing rates are proportional to spatial gradients (Smyth and Moum, 2001).

MODEL STUDY

A series of model studies over the discharges (62 and 77 lit/s), $P=36^{cm}$ and tailwaters (150-300 mm) were conducted by Ilkhanipour. A rectangular flume with a horizontal floor, 1^{m} wide, 7^{m} length and 0.60^{m} deep was used in the investigation. The flume side walls were Plexiglass sheets. Numerous experiments were performed by locating the orifice on horizontal, inclined and vertical planes 6^{cm} above the flume bottom in side by side with the place of lower nappe impact on flume bottom (on discharge equal 77 lit/sec) in an effort to obtain the best angle of orifice position plane(Figure 1). Orifice dimensions were 93.5 × 3 cm occupying the width & length of the flume respectively. The orifice had two spans relatively similar to crest two spans between the middle pier and abutments. The pier and abutments are not shown in Figure 1.

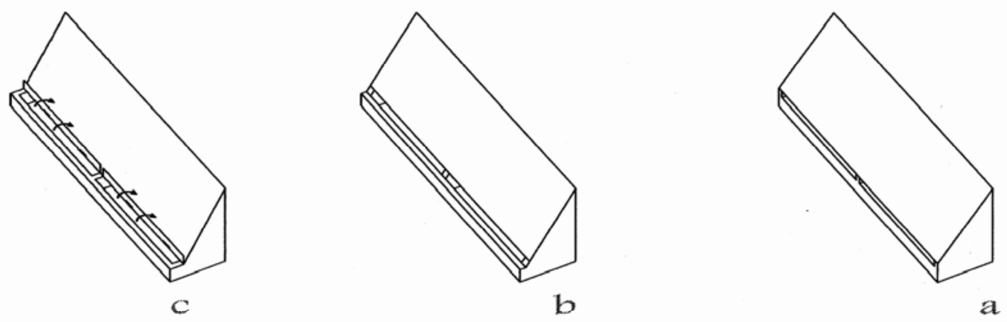

Figure 1: Ilkhanipour model with orifice location on a) vertical plane; b) inclined plane; c) horizontal plane with upstream-hinged gate on it

CONCLUSION

Just before the collision, according to Euler equation, tangential acceleration component will be equal for two jets flow. The gravitational and external forces acting on the nappe of water flowing over the weir are almost in the opposite directions whereas the gravitational and external forces acting on the jet flow from orifice are almost in the same directions. Therefore, after first collision, the accelerations of motion of the jet flow from orifice and the nappe of water flowing over the weir are increased and decreased respectively. Also, due to first collision, the curvature of the nappe of water flowing over the weir is changed and the curvature of the jet flow from orifice is increased. First collision create a large eddy structure, which rotates in the anticlockwise direction (left side view) on the lower face of the jet flow from orifice due to its roll-down as a result of collision. The outflow from orifice is divided to two parts, downward flow and backward flow. Trailing vortex is created between the downward flow and the nappe of water flowing over the weir. Therefore, a bubble is formed between two jets flow. Schematic and photographs of two jets flow attack induced-turbulence zone are shown in figures 2, 3. .

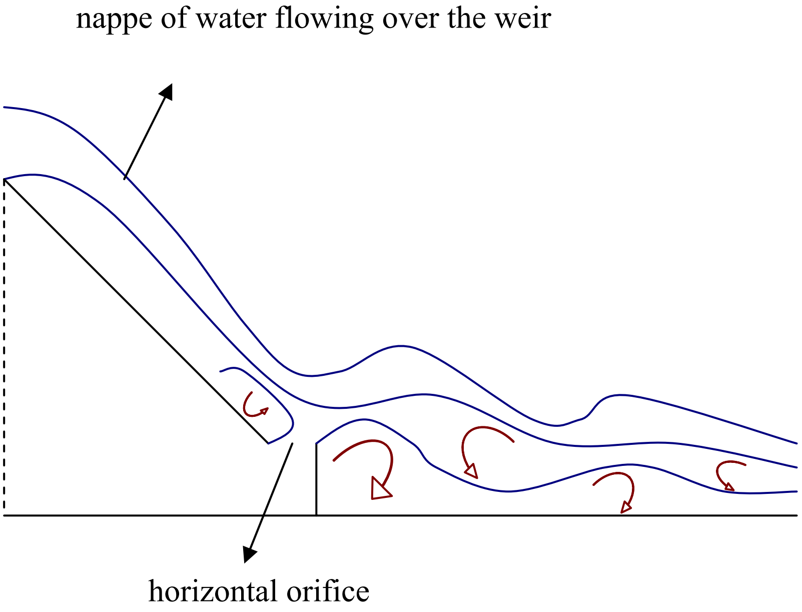

Figure 2: Schematic of eddies formation mechanism

Figure 3: Photograph of the boat shaped waffle floor flow pattern for condition of without any tailwater at the downstream of weir

Backward flow increase with an increase in the angle of attack. Backward flow will rotate in clockwise direction. The temperature of the backward flow water is increased due to collision and conversion of the kinetic energy to heat. Just before the place of collision, backward flow water temperature will be more than the temperature of water flowing over the weir. Backward flow cuts the nappe of water flowing over the weir alternately, and exit from the back of the nappe of water flowing over the weir due to the temperature gradient. This action causes scattering and other random collisions with other fluid particles which create air-water flow regions with high vortices. The boat shaped waffles are formed between two trailing vortexes. Trailing vortexes merge in the direction of fluid motion. Permanent flow velocity in high vortices region is generally lower than that in the boat shaped waffle region. Because of this velocity gradient, a shear layer is created at the interface of two regions (Ali, Hosoda and Kimura, 2007). The shear layer causes momentum transfer in the lateral direction from boat region to the two sides' high vortices regions. Two jets combined flows will have boat shaped waffle floor pattern with zigzag waffle and damped oscillations in the direction of motion Figure 4. Around the waffles contain air-water regions with high vortices. Photograph of flow pattern for condition of some tailwater existence at the downstream of weir is shown in Figure 5.

Figure 4: damped oscillations flow pattern in the direction of motion

Figure 5: Flow pattern for condition of some tailwater existence at the downstream of weir

Experimental tests results in term of hydraulic jump distance from the crest of weir, confirmed the presented theory to this type of stilling basin and it was found that the orifice should be located on horizontal plane, to increase the angle of collision and energy dissipation due to two jets flow attack-induced turbulence Figure 6 (Ilkhanipour, 2007).

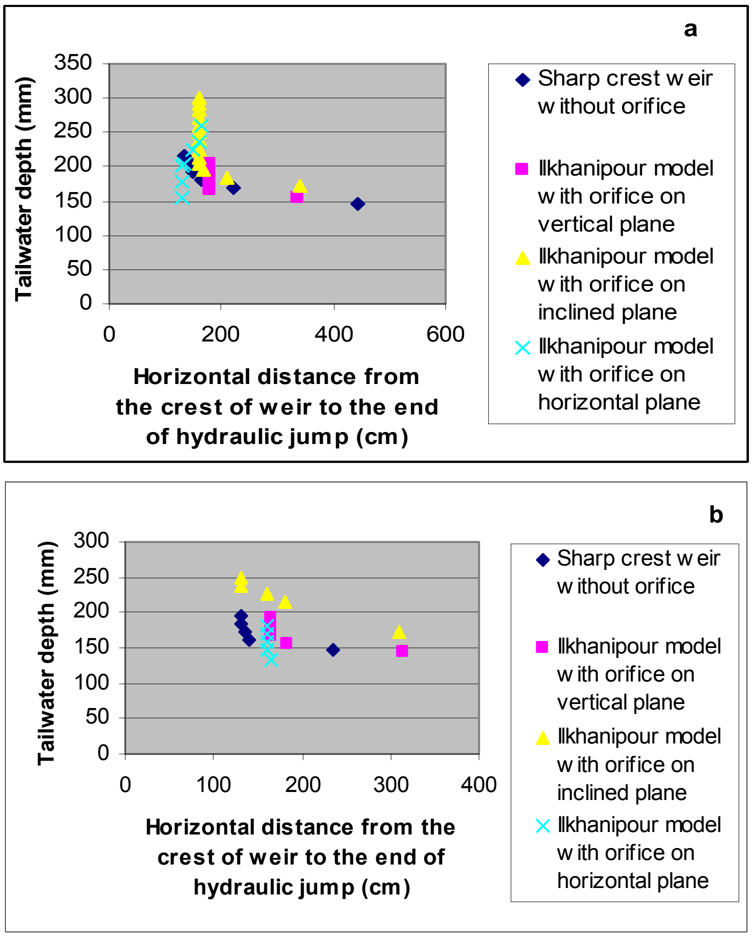

Figure 6: The effects of the orifice outflow inclination on horizontal distance from the crest of weir to the end of hydraulic jump at different tailwaters
a) Q= 77 lit/sec; b) Q= 62 lit/sec

REFERNCES

Ali, M. S., Hosoda, T. and Kimura, I. (2007), Effect of empirical functions of non-linear k-ε model on the prediction of flow resistance in a compound channel, In proceedings of the international conference on water and flood management, Vol. 2, pp. 735-743.

Alonso, M. & Finn, E. J. (1967), Fundamental University Physics, Vol. 1, Mechanics, Addison-Wesley, Massachusetts, USA

Ahmed, S. R. (1983), Influence of base slant on the wake structure and drag of road vehicles, Journal of fluid engineering, Vol. 105, pp. 429-434.

Bhagwat, M., Leishman, G. (2002), Generalized viscous vortex model for application to free-vortex wake and aeroacoustic calculations, 58th Annual forum and technology display of the American helicopter society international, Montreal, Canada, June 11-13.

Burgers, J. M. (1948), A mathematical model illustrating the theory of turbulence, Advances in applied mechanics, Vol. 1, pp. 171-199.

Farge, M. and Schneider, K. (2001), Coherent vortex simulation (CVS), a semi-deterministic Turbulence model using wavelets, Flow, turbulence and combustion, 66: pp. 393-426.

Fiorotto, V., Rinaldo, A. (1992), Turbulent Pressure Fluctuations under Hydraulic Jump, IAHR Journal of Hydraulic Research, 30(4), 499-520.

Fox, R. W., Macdonald, A. T. (1994), Introduction to fluid mechanics, John Wiley & Sons.

Gonzalez, C. and Chanson, H. (2005), Experimental study of turbulence manipulation in Stepped spillways implications on flow resistance in skimming flows, Seoul, Korea, pp. 2616- 2626.

Ilkhanipour Zeynali, R. (2007), Ilkhanipour model for diversion dams, In proceedings of the International conference on fluid control, measurements and visualization, CD-ROM Paper No. 10.

Mortazavi, I. and Giovannini, A. (2001), The simulation of vortex dynamics downstream of a Plate separator using a vortex-finite element method, International Journal of Fluid Dynamics, 5(4), pp. 41-58.

Newman, B. G. (1959), Flow in a viscous trailing vortex, The aeronautical quarterly, Vol. 1, pp. 167-188.

Ogawa, A. (1993), Vortex flow, CRC series on fine particle science and technology, CRC Press.

Smyth, W. D. and Moum, J. N. (2001), 3D turbulence, college of oceanic & atmospheric sciences, Oregon state university, Corvallis, OR 97330, USA, Academic press.

Vatistas, G. H. (1998), New model for intense self-similar vortices, Journal of propulsion and Power, Vol. 14, No. 4, pp. 462-469.

Vatistas, G. H. (1991), A simpler model for concentrated vortices, Experiments in fluids, Vol. 11, pp. 73-76.