

Prediction of flow patterns in local scour holes caused by turbulent water jets

Prédiction de configurations d'écoulement dans les cavités d'affouillement local causées par les jets d'eau turbulents

O.A. KARIM, *Lecturer, Department of Civil Engineering, University of Kebangsaan, Malaysia*

K.H.M. ALI, *Senior Lecturer, Department of Civil Engineering, University of Liverpool, United Kingdom*

ABSTRACT

The paper describes a testing procedure for investigating the suitability of the FLUENT CFD package in simulating flow patterns, generated by a turbulent water jet impinging on rigid horizontal and scoured beds. The scoured beds were formed at various times of the scouring process. The three turbulence closure models incorporated in FLUENT are: the Standard k - M Model, the Reynolds Stress Model and the ReNormalisation Group Theory-Based Model. Results obtained using these models were examined. In general, flowfields, velocities and shear stresses predicted by FLUENT showed close agreement with relevant experimental results.

RÉSUMÉ

Cet article décrit une procédure de test pour évaluer la capacité du progiciel de mécanique des fluides FLUENT à simuler les écoulements générés par un jet d'eau turbulent venant heurter des fonds fixes horizontaux et affouillés. Ces fonds correspondent à différents instants du processus d'affouillement.

Les trois modèles de fermeture turbulente inclus dans FLUENT sont: le modèle k - M standard, le modèle de tensions de Reynolds et le modèle basé sur la Théorie de reNormalisation de Groupe. Les résultats fournis par ces modèles sont analysés.

En général, les écoulements, les vitesses et les contraintes de cisaillement prédites par FLUENT sont en bon accord avec les résultats expérimentaux correspondants.

Introduction

Prediction of local scour holes that develop downstream of hydraulic structures plays an important role in their design. Excessive local scour can progressively undermine the foundation of the structure. Because complete protection against scour is too expensive, generally, the maximum scour depth and the upstream slope of the scour hole have to be predicted to minimise the risk of failure.

The localised scour phenomenon has been the subject of extensive investigations by many researchers and numerous literature exists for scour caused by 2 and 3 dimensional turbulent jets. Most of the studies conducted on scour have been empirical because of the complexity of the physical processes.

The pioneering investigation on scour due to a jet was done by Rouse (1939). Scour due to a horizontal wall jet was studied by Laursen (1952). Scour by circular impinging jets was studied by Doddiah et al (1953), Poreh and Hefez (1967), Sarma and Sivasankar (1967), Westrich and Kobus (1973), and Rajaratnam and Beltaos (1977). Iwagaski et al. (1958) undertook an analytical study of scour caused by a three-dimensional jet. Carstens (1966) developed an empirical formula for sediment transport rate by analysing the experimental data of Laursen (1952). Scour caused by impinging plane jets was studied by Altinbilek and Okyay (1973) and by Francis and Ghosh (1974). Scour by circular and rectangular turbulent wall jets was studied by Rajaratnam and Berry (1977) and Rajaratnam and Humpries (1984) respectively. The scour due to plane wall jets in shallow tailwater was examined by Rajaratnam and MacDoughall (1983). Hassan and Narayanan (1985) investigated local scour downstream of rigid aprons. Local scour caused by a submerged wall jet was studied by Ali and Lim (1986). Uyumaz (1988) investigated scour patterns

downstream of vertical gates. Mason (1988) studied plunge pool scour. A study of the scour pattern in shallow tailwater was done by Johnston (1990). Ali and Salehi-Neyshabouri (1991) and Ali and Whalley (1992) conducted experiments on local scour downstream of offset jets.

The studies of the aforementioned investigators have made important contributions to the knowledge of the phenomena of local scour downstream of hydraulic structures in the relevant flow situations.

In practise, nearly all jets of importance are turbulent in nature. This would normally require expensive and sophisticated instrumentations in order to be able to measure the quantities desired. Since the computer capability has drastically increased numerical simulation seemed a promising means as an alternative. Furthermore, the flexibility in defining the geometry in the numerical analysis enables various design options to be considered at a relatively fast time and lower cost. The wealth of experimental data available in the literature makes it possible to validate the numerical simulation before it can be used more confidently. The purpose of this paper is to verify the effectiveness of the numerical routine incorporated in the FLUENT package. In this paper the validation begins with the simulation of flows in rigid and scoured beds caused by turbulent water jets. Holes produced at various times from the start of the scouring process were "frozen" by Ali and Lim (1986) using sheets of thin aluminium foil to cover the whole bed including the hole. The surface of the foil was then coated with the sand grains used in the experiment in order to simulate the original roughness of the bed. A check on the centreline bed profiles before and after the foil showed agreement in bed levels to within 5%. Two categories of submergence of water jet were selected:

Revision received August, 1999. Open for discussion till February 28, 2001.

- a) The deeply submerged jet where the tailwater depth, H , is many times the height, y_0 , of the jet opening ($H \gg y_0$), and
- b) The jet with minimum submergence where H is approximately equal to the height of the jet opening ($H \approx y_0$).

In the following section, a brief review of the mechanism of the scour by water jets is presented. It must be stressed that all the scour holes were produced in non-cohesive sediments.

Local Scour Caused by Turbulent Water Jets

The entrainment, transportation and deposition of sediment depends as much on the properties of the sediment as upon the hydraulic characteristics of the flow. Generally speaking, the scouring mechanism involves a combination of two relative motions: that of the flow field relative to the boundary, and that of the bed material relative to the overlying flowing fluid.

In general, localised scour can occur in two ways (1) clear-water scour; and (2) live-bed scour. Clear-water scour refers to the situation where no sediment is supplied from upstream into the scour zone. Live-bed scour, on the other hand, occurs when there is general bed load transport by the stream. Sediment is continuously being supplied to the areas subjected to scour. The present analysis deals mainly with the clear-water scour.

The Classical Wall Jet (CWJ)

Since the turbulent wall jet is the scouring agent investigated in the present study, it is useful to introduce the general characteristics of the CWJ first. The wall jet is defined as a jet of fluid, impinging tangentially (or at angle) on a boundary surrounded by stationary (or moving fluid). The case of the classical wall jet, i.e. the plane turbulent wall jet issuing into the same stationary fluid of semi-finite extent on a rigid boundary, is shown in Figure 1.

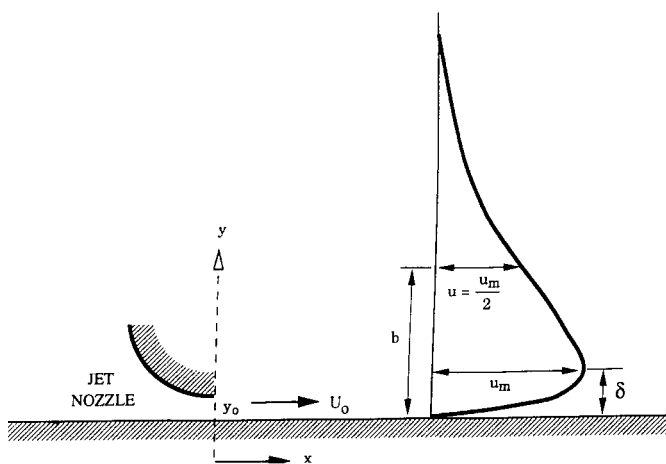


Fig. 1. Definition sketch for the classical wall jet.

Experimental investigations on the plane turbulent wall jet under zero pressure gradient have been studied extensively, and a comprehensive bibliography on wall jets can be found in the book by Rajaratnam (1976). More recent work on turbulent jets was carried out by Altinbilek and Okay (1973), Lim (1985),

Salehi-Neyshaboury (1990), Ali and Salehi-Neyshaboury (1991) and Ali and Walley (1992). It was found that for sections located at a longitudinal distance x greater than 15 times the thickness of the jet opening, y_0 , the transverse velocity plotted in dimensionless co-ordinates is similar for all sections. For these similar velocity distributions, the non-dimensionalised velocity, known as the velocity scale, is the local horizontal velocity divided by the maximum velocity at that section, u_m . The corresponding length scale is the distance normal to the direction of u , b , from the boundary plane at which $u = u_m/2$ and the velocity gradient, du/dy is negative.

Description of the Mechanism of Scour by Water Jet

Consider a turbulent jet issuing from a 2-D or 3-D nozzle. As the diffused jet starts to flow over the initially flat erodible bed, the scouring potential of the jet, created by the high incoming velocity, is usually so strong that the sand grains are immediately dislodged from the surface and transported downstream at a rapid rate. For a small time period, the vertical dimensions of the scour hole increase at a faster rate than the horizontal dimensions and the bed material is transported mainly as bed loads.

As the scour hole gradually enlarges with time, the depth of water also increases. From the principle of continuity, the expanding cross sectional area would require a reduction in the mean flow velocity. Accordingly, the local velocity near the scoured bed decreases as the depth of the hole increases. Hence, the rate of erosion decreases as time progresses. For larger time periods, the bed velocity eventually reduces to a certain "critical" value whence the flow (near the bed) becomes incapable of removing further bed material from the scour area. At this point, we could say that the scour geometry has reached its "equilibrium" or "asymptotic" condition. Implicitly, this suggests that the local bed velocity is directly related to the concept of a threshold of a particular movement.

It has been accepted that the boundary shear stress is a logical choice of a key variable to characterise the incipient motion of particles for a given bed slope. The bed shear stress is a function of the local bed velocity and the friction factor, which is often assumed to be constant, therefore, measurements of the bed velocity and distributions in a scour hole will enable the scouring process to be examined.

Description of the Flow Patterns in Local Scour Hole

Interesting observations were made by Rajaratnam and Berry (1977) regarding the velocity distributions (using an air jet) along scour hole at the asymptotic state. They noted that the jet behaves like a free jet up to the section of maximum erosion and from this section up to the ridge section, it behaves like an obliquely impinging jet, developing wall-like velocity profiles in the vicinity of the ridge. Figure 2 shows a typical plot of Rajaratnam and Berry's velocity distributions. The velocity results showed that up to approximately the section of maximum erosion, the variation of u_m/U_0 was well described by the simple circular jet equation:

$$\frac{u_m}{U_o} = 6.67 \left(\frac{y_o}{x} \right) \quad (1)$$

in which u_m is the maximum velocity of the velocity profile at any x distance from the nozzle, y_o is the jet thickness, and U_o is the average jet velocity. The velocity profiles for this region in the vertical and horizontal planes (through the maximum velocity points) were found to be similar (see Figure 3) and the data were described well by the exponential equation:

$$\frac{u}{u_m} = e^{-0.693(y/b)^2} \quad (2)$$

and

$$b = 0.097x \quad (3)$$

where y = vertical distance from the axis of jet at any x direction, and b = vertical distance from the axis to the point where $u = u_m/2$. Near the bed, the velocity distribution was modified because of the no-slip condition.

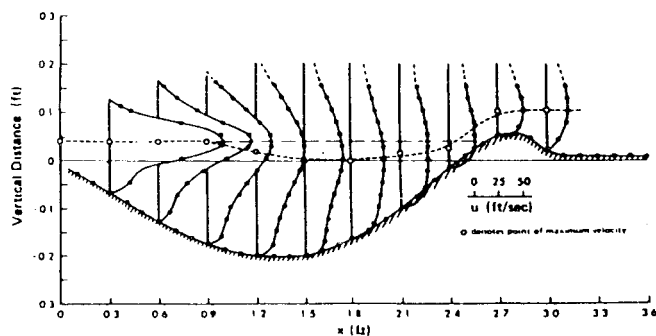


Fig. 2. Velocity distributions generated by a three-dimensional air jet at the asymptotic state (after Rajaratnam & Berry, 1977).

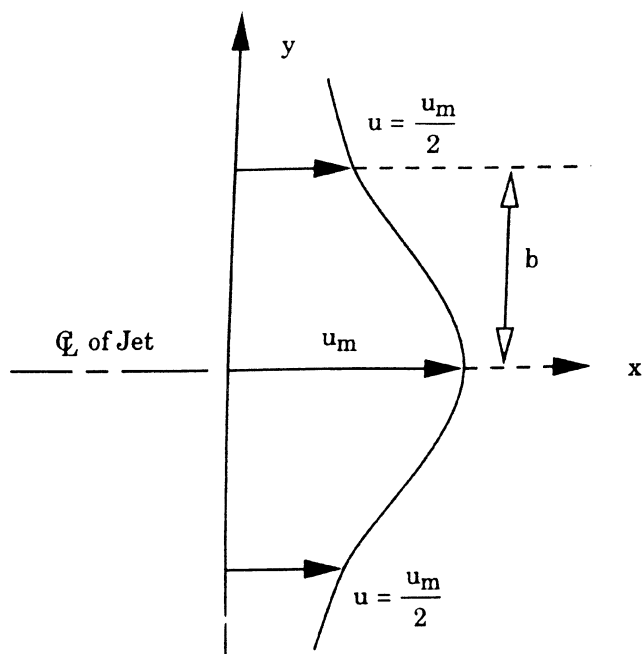


Fig. 3. Definition sketch for the velocity distribution of a typical free jet.

Theoretical computation of velocity profiles in the scour hole was attempted by Breusers (1975), based on the method of Rosinski (1961). This method was also used by Ali and Lim (1986) to predict the velocity distribution in a scour hole caused by 2-D turbulent wall jets. Ali and Lim (1986) performed an extensive investigation into the hydrodynamic behaviour of 2-D and 3-D turbulent wall jets with minimum tailwater depth ($H \approx y_o$).

The FLUENT CFD Package

FLUENT is a multi-purpose computer software for modelling fluid flow, heat transfer and chemical reaction which enables a rapid analysis of complex flows. FLUENT applies computer simulation methods to analyse and solve practical design problems based on fundamental principles of computational fluid dynamics (CFD) such as the conservation of mass, momentum and energy. FLUENT (1993) is a two part program consisting of a pre-processor, PreBFC V4, and a main module, FLUENT. PreBFC V4 was used to define the geometry and a structured grid of the problem to be modelled. The grid information was then imported from PreBFC to FLUENT. The physical models, fluid/material properties, and boundary conditions that describe the problem to be modelled were next added to the grid information and stored in Case-File that is a record of all the inputs for problem definition. The results of the calculations were stored in Data-File (Karim, 1996).

Laminar and turbulent flows are governed by the laws of mass, momentum and energy conservation. Modelling of turbulent flows requires appropriate modelling procedures to describe the effects of turbulent fluctuations velocity and scalar quantities on the basic conservation equations for laminar flow. FLUENT uses the standard Reynold's averaging of the conservation and momentum equations (Karim, (1996)). In order to close the resulting system, some of the correlations are usually approximated in terms of quantities that can be calculated. "Model" assumptions about turbulence are thereby introduced which may not always be realistic. Instead of the real turbulent fluid we thus consider an idealised "model" fluid which is governed by the laws prescribed in a "turbulence model."

The main task of a turbulence model is to provide expressions or closure models that allow the evaluation of the correlations of the unique expressions in terms of mean flow quantities (Karim, (1996)). FLUENT uses the following turbulence closure models:

- i the Standard $k-M$ model
- ii the Differential Reynold's Stress Model (RSM), and
- iii the ReNormalization Group (RN6) $k-M$ model. Details of these methods are given in Fluent (1993) and in Karim (1996).

The choice of a particular scheme is not a clear cut issue. Some judicious experimentation of the relative stability characteristics of both schemes for a given number of iterations may be required before picking one of the schemes. The default solver or discretization scheme used was the Power-Law Differencing

Scheme. Alternatively, a higher order scheme termed the blended Second Order Upward (Central Difference scheme was also applied in order to investigate the significance of the scheme. While the higher order scheme was supposed to provide improved accuracy, it was found that numerical instability occurred during the iterations. For that reason, the Power-Law scheme was chosen for the present analysis.

If the normalised residuals were converging stably but at a slower rate, the underrelaxation factor can be increased for the velocity, turbulence kinetic energy and eddy dissipation, to speed up the convergence. However, this has to be done carefully as an increase in the underrelaxation factors may also lead to a sudden divergence. An increase in the underrelaxation factors will only have an effect on the speed of convergence and not on the final converged solutions of the flow fields. Similarly, the use of pressure-velocity coupling algorithms in FLUENT, either SIMPLE or SIMPLER, will give the same solutions when converged as both methods are only corrections of the variables. The question of which method to use only influences the convergence speed. In this paper, the default algorithm i.e. SIMPLE was used. A trial run using SIMPLER did not exhibit any significant increase in the convergence speed.

Choosing a Discretization Scheme in FLUENT

The process of obtaining a converged solution is of great importance in the numerical simulation procedures. The residuals or normalised residuals are a measure of the degree to which each equation is satisfied throughout the flow field. Generally, a solution is well converged when the normalised residuals are of the order of 1×10^{-3} . If the residuals have decreased to this level, are monotonically decreasing, and the flow field looks unchanged from the solution so iterations earlier, then the solution can be considered "converged".

Model validation

As mentioned earlier, the aim of this paper is to verify the effectiveness of the numerical procedure available in FLUENT in predicting the flow characteristics of turbulent wall jets. For that purpose velocity measurements carried out by Wu and Rajaratnam (1995) and Ali and Lim (1986) were used as benchmarks. The analysis were divided into three sections.

- Flat bed – deep submergence (Wu and Rajaratnam)
- Flat bed – shallow submergence (Ali and Lim, $t = 0$) where t = time from commencement of scouring action
- Scoured bed - shallow submergence (Ali and Lim, $t = 15, 90$ and 400 mins.)

These times correspond to dimensionless ratios Ut/y_0 of 9.71×10^3 , 5.82×10^4 and 2.59×10^5 respectively.)

The experimental set-up and procedures are briefly described in the following section in order to appreciate the domain to be modelled.

Wu and Rajaratnam's Experiments (1995)

The experiments were conducted in the T. Blench Hydraulics Laboratory of the University of Alberta. The submerged jets were produced in a flume, 7.60 m long, 0.466 m wide and 0.60 m deep. The tailwater depth was controlled by a vertical gate located at the downstream end of the flume. Water entered the flume under a sluice gate with a streamlined lip. The time-average velocity field was measured with either a Pitot probe or a Prandtl probe.

Four experiments were conducted and are summarised in the following table:

Table 1. Summary of Wu and Rajaratnam's Experiments.

| Run | H(m) | U_o (m/s) | F_o |
|-----|------|-------------|-------|
| 1 | 0.44 | 1.72 | 5.48 |
| 2 | 0.53 | 2.86 | 7.46 |
| 3 | 0.46 | 2.86 | 7.46 |
| 4 | 0.39 | 2.86 | 7.46 |

where: y_o = thickness of jet

H = tailwater depth

U_o = uniform initial jet velocity

F_o = Froude number = $U_o/\sqrt{gy_o}$

In these experiments, the gate and hence and depth of the supercritical stream leaving the gate was equal to 10 mm for Run 1 and 15 mm for Runs 2, 3 and 4 and 15 mm. To ensure that the flow was two dimensional, they measured the velocity field in the central vertical plane (CVP) and another vertical plane located at a transverse distance of 0.1 times the width of the flume from the CVP. Their comparison of the longitudinal velocity profiles showed that at least in the central part of the flume, the flow was two dimensional.

Ali and Lim's Experiments (1986)

The experimental set-up for the sluice-gate experiments is given by Ali and Lim (1986). A long channel was used which had the dimensions of 5.0 m long, 0.61 m wide and 0.5 m deep. A well streamlined sluice gate was constructed in such a way so that it ensured uniform jet flow (Ali and Lim, (1986)). Sand was filled to a height of 30 cm in the working section. At the end of the channel, a sluicing arrangement allowed the tailwater depth to be regulated.

The bed material used was uniform sand with the following characteristics: $d_{50} = 0.82$ mm, $\rho_s = 1.126$, specific gravity of 2.66, angle of repose $\theta = 33^\circ$, and, porosity of 0.403.

At the end of the experiment, measurements of the scour hole were made using a depth gauge. Figure 4 shows the time evolution of centreline bed-profiles obtained with tailwater depth, $H = 6.7$ cm. For present model verification analysis a period of $t = 0$ (initial bed), 15, 60 and 400 (asymptotic) were simulated ($U_o t/y_0$ of 9.71×10^3 , 5.82×10^4 and 2.59×10^5).

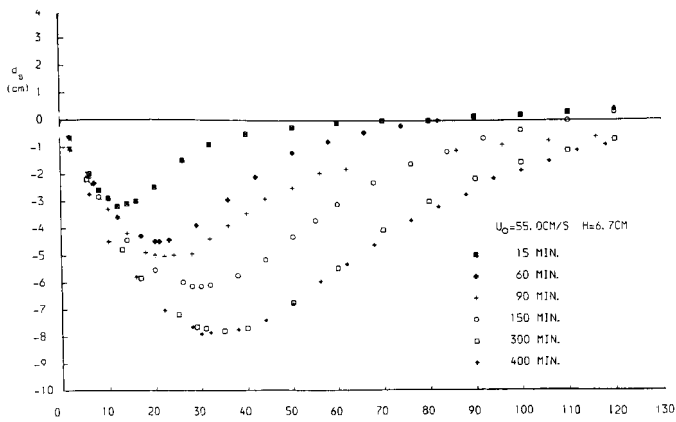


Fig. 4. Time evolution of centreline bed profile (after Ali and Lim, 1986). Waterdepth = 67 mm, $U_0 = 0.55$ m/s, $t = 15, 60, 90, 150, 300$ and 400 minutes. Corresponding $U_0 t / y_0$ values of $9.74 \times 10^3, 3.90 \times 10^4, 5.8 \times 10^4, 9.74 \times 10^4, 1.95 \times 10^5, 2.60 \times 10^5$.

The time development of scour represents a very complicated flow phenomenon involving the movement of sediment-water mixture along the bed of the scour hole. Because of the continuously changing flow boundary as scour progresses with time, the velocity distribution pattern in the scour hole would likewise be different at different stages of the erosion process. In order to investigate these transient non-uniform flow patterns, a steady-state flow model was necessary. (Lim (1985) converted the loose sand scour hole into a fixed-bed model so that the flow characteristics in the hole at that instant of time could be studied in detail. Lim achieved this by covering and moulding the scour hole *in-situ* with very thin aluminium foil. The foil was coated with the sand grains used in the experiment. The velocities were measured using a miniature streamflow current meter. The measuring hand of the current meter was 16 mm in diameter. The “floor velocity” was measured at $y = 0.8$ cm (equal to half of the diameter of the streamflow meter) above the bed of scoured hole. Since the probe only records positive velocity, reversed flow was judged by observing the direction in which the rotor turned. Based on a measured “floor” velocity, they calculated the boundary shear stress using a method developed by Melville and Raudkivi (1977):

$$\tau_b = \frac{\rho f u_b^2}{8} \quad (4)$$

where τ_b = shear stress near the bed ρ = fluid density; u_b = velocity near the bed and f = friction factor obtained using the Colebrook-White equation:

$$\frac{1}{f^{1/2}} = -2 \log_{10} \left(\frac{k_s}{14.83 H_T} + \frac{2.52}{f^{1/2} R_H} \right) \quad (5)$$

k_s is the height of the sand roughness; and $H_T = u_b H_T / \nu$.

Simulation Using Fluent

The experiments mentioned in the above section were simulated using the FLUENT package. Description of the package and the governing equations are given in Karim (1996). FLU-

ENT V4 is a two part program consisting of a pre-processor, PreBFC V4, and a main module PreBFC V4 was used to define the geometry and a structured grid of the problem to be modelled. The grid information was then imported from PREBFC to FLUENT. The physical models, fluid/material properties, and boundary conditions that describe the problem to be modelled were next added to the grid information and stored in Case-File that is a record of all the inputs for problem definition. The modelling steps are summarised as follows:

Import the PreBFC Grid File; Choose the basic equations; Set the physical constants; Calculate the solution; Examine the results; and Save the results.

These steps are described in detail by Karim (1996).

Results and discussion

General

This section reports the results obtained from the numerical simulation. The FLUENT’s results are compared with the selected benchmarks experimental results. A typical physical grid for one of Ali and Lim’s experiments is shown in Figure 5. The three closure models available in FLUENT were tested using scoured bed profile ($t = 400$ mins. $U_0 t / y_0 = 2.59 \times 10^5$) obtained by Ali and Lim. Comparisons of the results are presented in Figure 6. There are no significant variations in the computed values between the Standard k - M model and the RNG k - M model. However, as shown in Table 2, the RNG model converges twice as fast as the other models, thereby increasing the computational efficiency. The RSM model yielded good agreement except at sections beyond that of maximum scour.

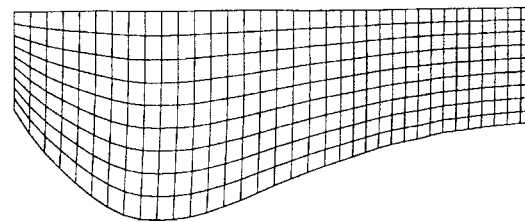


Fig. 5. Physical grid arrangement for a scour hole.

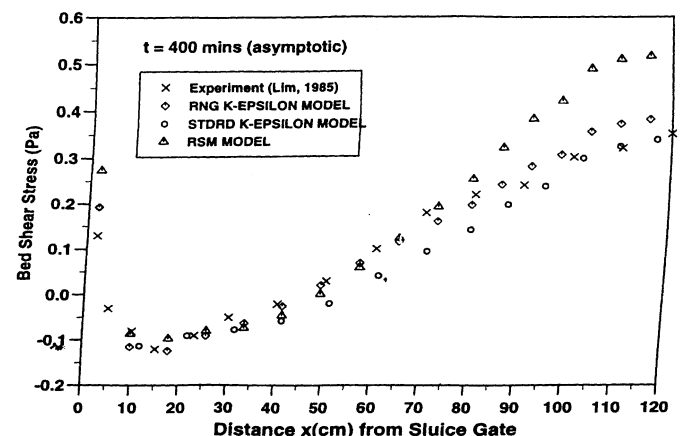


Fig. 6. Comparison between Ali and Lim’s (1986) experimental bed shear stresses and FLUENT results using three turbulence models ($t = 400$ minutes, $tU_0/y_0 = 2.60 \times 10^5$.)

Table 2. Comparison of converging time between different closure model in FLUENT ($t = 400$ mins; $U_0 t / y_0 = 2.59 \times 10^5$).

| CLOSURE MODEL | Number of iterations to converge |
|-----------------------|----------------------------------|
| Standard \square :- | 1225 |
| RNG \square :- | 705 |
| RSM | 763 |

Study was also made to see the effect of different grid sizes. The grid sizes used are given in the following table.

Table 4. Grid sizes used in the study ($t = 400$ mins; $U_0 t / y_0 = 2.59 \times 10^5$).

| RUN | Grid Nodes |
|-----|------------|
| 1 | 71 x 11 |
| 2 | 140 x 11 |
| 3 | 71 x 20 |
| 4 | 140 x 20 |

In the finite difference solution, the absolute size of cells within the grid is an important issue, since if the cells are too large, the solution obtained can be dependent on the cell size rather than purely on the physical constraints of the solution domain and the input conditions.

Another issue in the sizing of grid cells is the detail with which, and the locations at which, output is required. It is helpful if the cell centres can be located close to experimental measuring sections. The results for different grid sizes used are shown in Figure 7. It can be seen that the predicted bed shear stress is lower near the sluice gate and higher further downstream for finer vertical grid (J direction). Grid arrangements in Run 1 and Run 2 gave better agreement with the calculated values of bed shear but due to the less number of grid nodes involved in Run 1, the number of iterations required to converge is less than Run 2. For this reason, grid arrangement in Run 1 was adopted for the rest of the analysis.

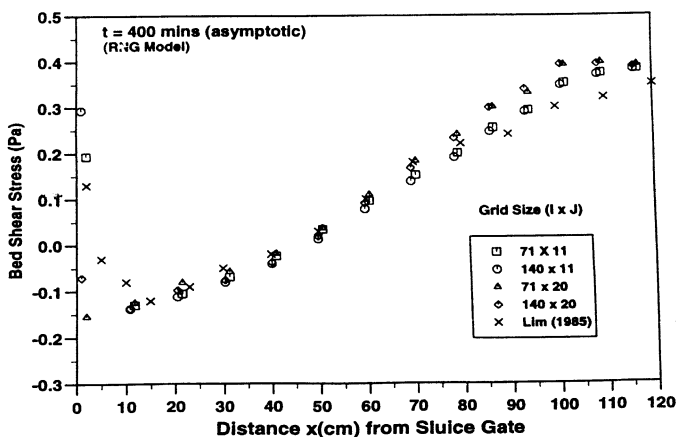


Fig. 7. Effect of grid size on the bed shear stress results ($t = 400$ minutes, $tU_0/y_0 = 2.60 \times 10^5$). Using the RNG turbulence model.

Flat-Bed Shallow Submergence

Figures 8a-8d show Ali and Lim's (1986) experimental velocity contours for a rigid bed ($t = 0$) and for scour holes produced after 15, 90 and 400 minutes.

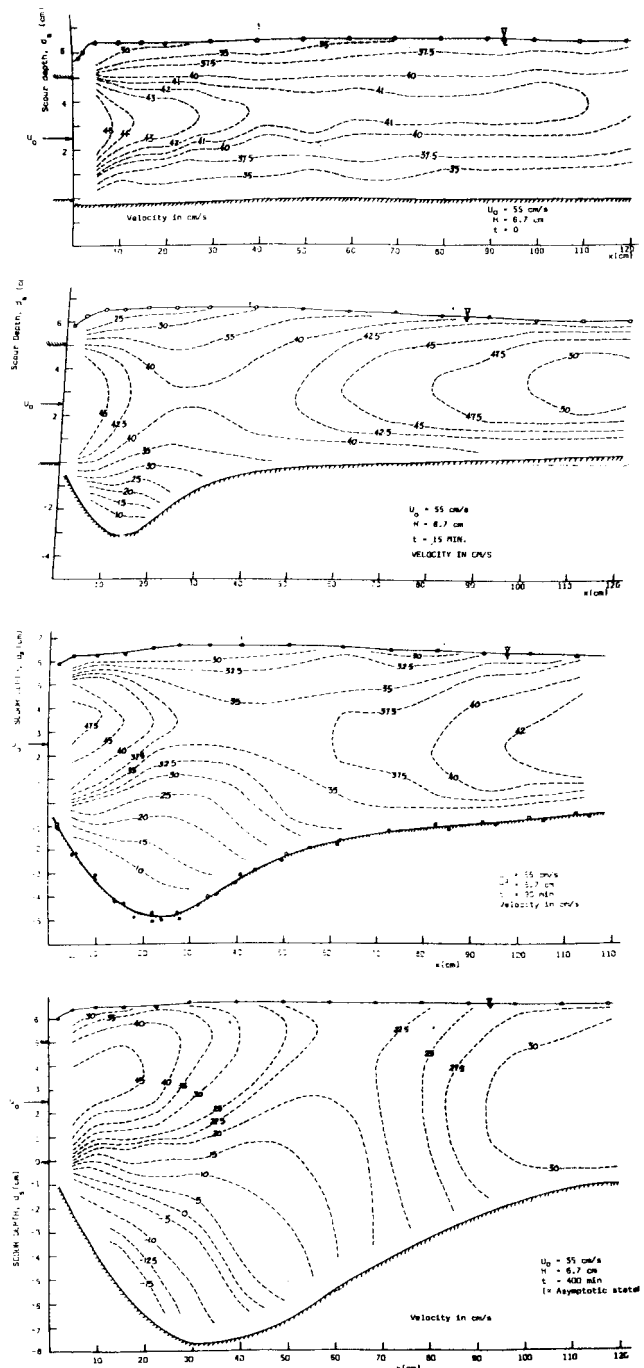


Fig. 8. Ali and Lim's (1986) experimental velocity contours.

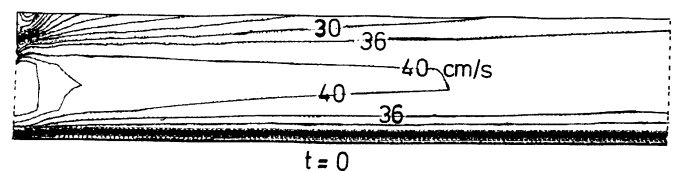


Fig. 9. FLUENT's predicted horizontal velocity contours for a rigid bed ($t = 0$ minutes).

Figure 9 shows the predicted FLUENT velocity contours at the initial flat bed conditions for shallow submergence case ($H \approx d_0$). Figures 8a and 9 show that the flow is very symmetrical about the centreline of the jet. The floor velocity calculated by FLUENT at 0.8 cm above the bed was 32 cm/s. This compared well with measured value obtained by Ali and Lim, which was 35 cm/s. Figure 10 shows a comparison between the bed shear stress obtained using FLUENT and Ali and Lim values (using Equation 1). Again the agreement is reasonably good with an average value of about 0.45 N/m².

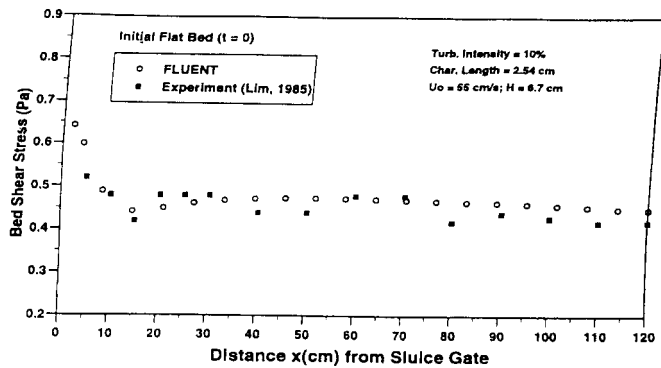


Fig. 10. Comparison between Lim's (1985) experimental bed shear stresses for a rigid flat bed and those predicted by FLUENT. Turbulence intensity = 10 %, characteristic length = 2.54 cm, downstream water depth = 6.7 cm and jet velocity = 0.55 m/s.

Scoured-Bed Shallow Submergence

Figure 8b shows that for $t = 15$ mins, the experimental floor velocities decrease rapidly from 35 cm/s to about 10 cm/s in the deepest part of the scour hole. As a result of the sudden expansion of the jet into the hole, the mean velocity of the jet decreases. Beyond the deepest part of the hole, the flow begins to converge with a corresponding increase in velocity. For $t = 400$ mins, Figure 8d shows that the line of maximum velocity was deflected towards the water surface and that negative velocities were present near the section of maximum scour.

Figures 11–13 show corresponding velocity contours predicted by FLUENT. In general, these results are close to the experimental distributions. Figures 14 and 15 show the velocity profiles and velocity vectors for $t = 400$ mins respectively. Reversed flow can be seen in these figures. These results confirm the observation made by Ali and Lim (1986), Rosinski (1961), Breusers (1967) and other researchers in their experiments. It can be seen in Figure 14 that the negative velocities (roller) ends in the section of maximum scour forming a boundary line between reverse and forward velocities in the bottom zone.

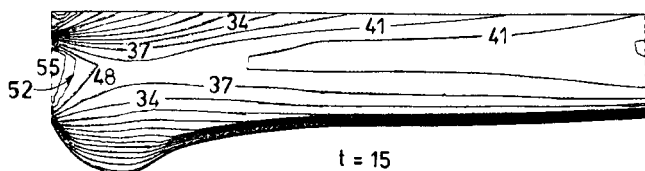


Fig. 11. Predicted horizontal velocities for $t = 15$ minutes ($tU_0/y_0 = 9.74 \times 10^3$).

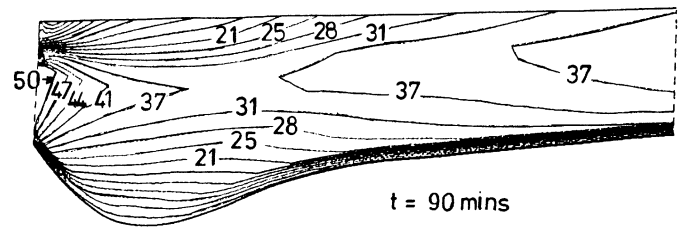


Fig. 12. FLUENT's predicted horizontal velocities for $t = 90$ minutes ($tU_0/y_0 = 5.85 \times 10^4$).

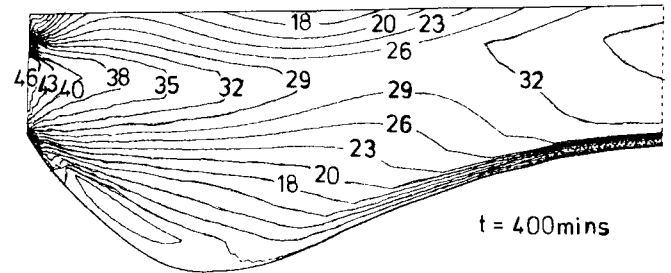


Fig. 13. Predicted horizontal velocities for $t = 400$ minutes ($tU_0/y_0 = 2.60 \times 10^5$).

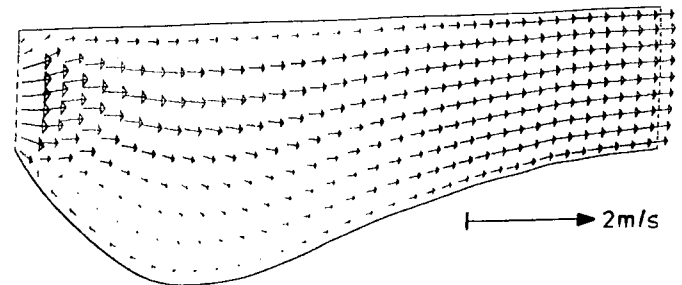


Fig. 14. Predicted velocities for $t = 400$ minutes ($tU_0/y_0 = 2.60 \times 10^5$).

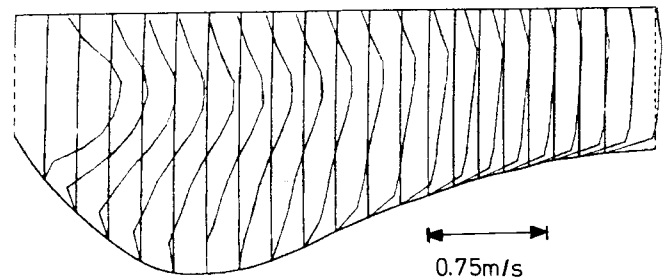


Fig. 15. FLUENT's predicted horizontal velocity distributions for $t = 400$ minutes.

Figure 16 shows the computed bed shear stress distribution for $t = 15, 90$ and 400 mins. The values, represent the x -components of the shear force at each wall-fluid interface divided by the area of that interface. These values are compared with the qualitative values computed by Ali and Lim (based on u_b at $y = 0.8$ cm from bed) and are shown in Figures 17–19. It can be seen that a good agreement was obtained between the values calculated from the experimentally measured "floor" velocities and those obtained from FLUENT.

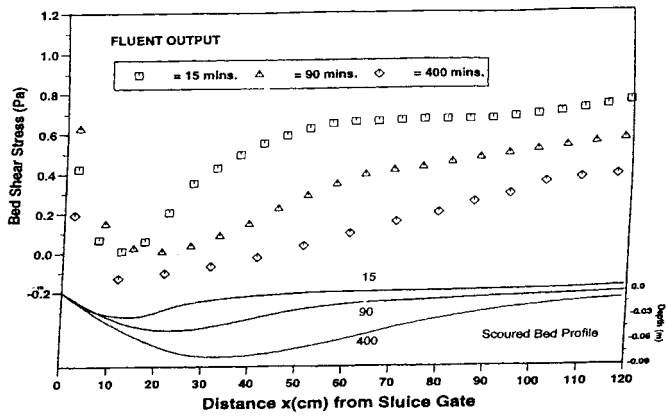


Fig. 16. Predicted bed shear stresses for various sizes of the bed scour holes.

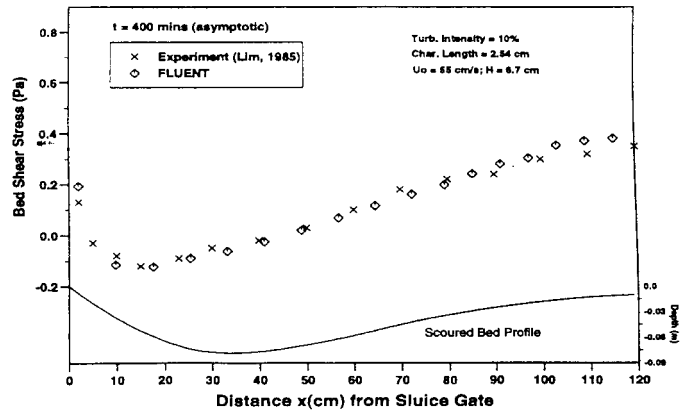


Fig. 19. Comparison between the experimental and FLUENT's predicted shear stress distributions for $t = 400$ minutes. ($tU_0/y_0 = 2.60 \times 10^5$)

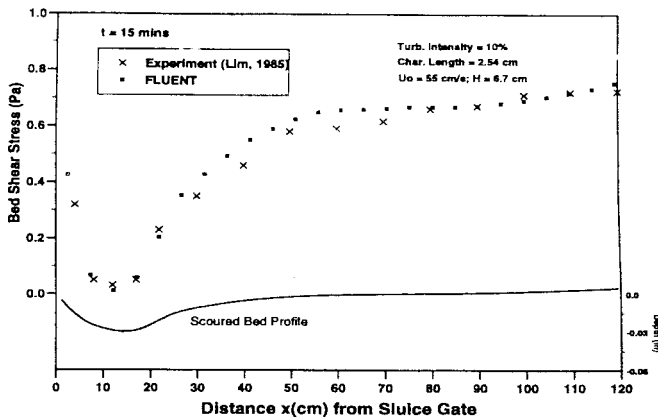


Fig. 17. Comparison between the experimental and the predicted shear stresses for $t = 15$ minutes.

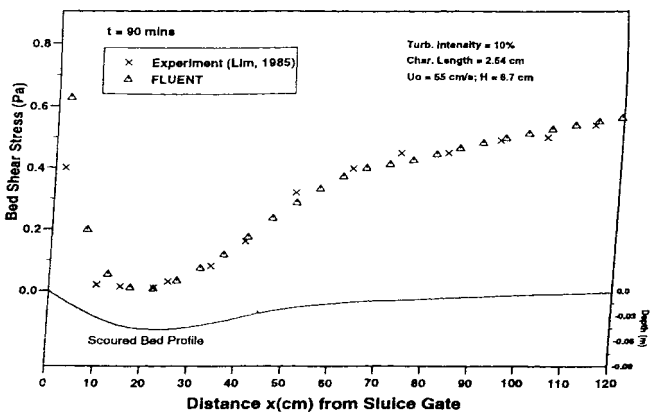


Fig. 18. Comparison between the experimental and the predicted shear stresses for $t = 90$ minutes.

Flat-Bed Deep Submergence

Wu and Rajaratnam plotted the longitudinal profiles in a non-dimensional form for each experiment using u_m and b as the velocity and length scales respectively (Figure 20). They observed that except for the sections very close to the nozzle (within the potential core region) and far away from the nozzle (near the end of the jump), the velocity profiles at all sections, in the forward flow region, are similar. FLUENT's results show the same observation and are given in Figure 21. Clearly, there is close agreement between the results of Wu and Rajaratnam and those obtained from FLUENT.

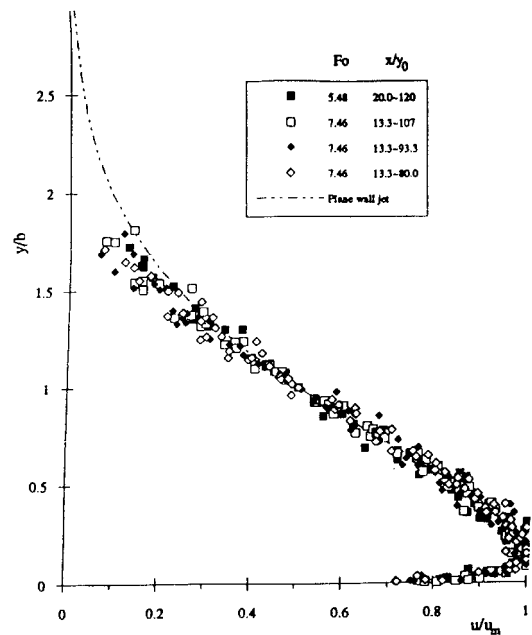


Fig. 20. Consolidated plot of the non-dimensional longitudinal velocity results of Wu and Rajaratnam.

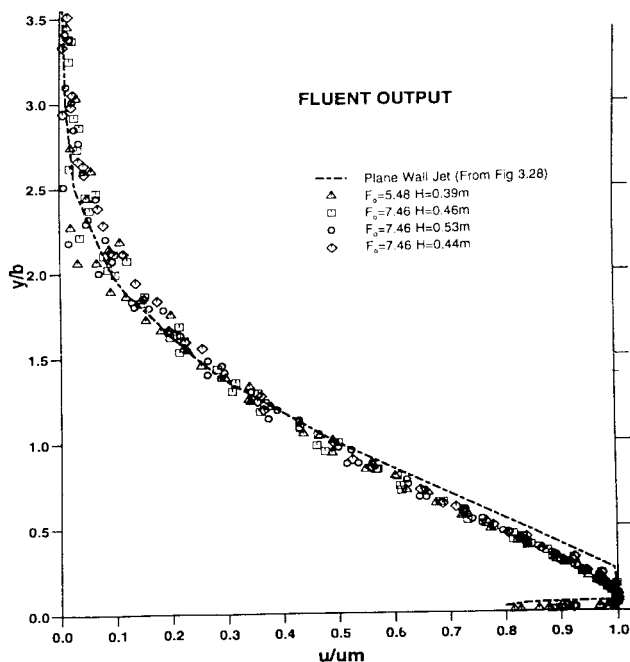


Fig. 21. Consolidated plot of the non-dimensional longitudinal velocity results predicted by FLUENT for Wu and Rajaratnam's (1995) Experiments.

Conclusions

1. The numerical prediction scheme provided by FLUENT was used to predict the 2-D flow velocity distribution and the bed shear stress for both flat bed and scoured bed. The computed values were then compared to the experimental values obtained by other researchers.
2. The FLUENT features allowed the basic turbulence equations and associated turbulence closure models to be solved with relative ease. The PreBFC and grid-generation facility in FLUENT enabled the use of variable finite-difference grid sizes so as to minimise demand for computer resources whilst still achieving useful simulation results. The use of Body-Fitted Co-ordinates gave better representation of the scoured-bed than the Cartesian Co-ordinates, resulting in more reliable bed shear stresses.
3. Overall, the computational results showed a close agreement with the various selected experimental results. In the case of bed shear stress, FLUENT results are consistent with the calculated results of Ali and Lim. The standard $k-M$ and the RNG $k-M$ models describe the flow at the boundary better than the RSM model. No significant difference in the results was observed between the $k-M$ and the RNG $k-M$, except that the RNG $k-M$ model converged twice as fast as the standard $k-M$ model.

References

- ALI, K.H.M. and LIM, S.Y. (1986). "Local scour caused by submerged wall-jets." *Proc. Institution of Civil Engineers*, London, England 81(2), pp 607-645

- ALI, K.H.M., and SALEHI-NEYSHABOURI A.A., (1991). "Localized scour downstream of a deeply submerged horizontal jet", *Proc. Instn. Civ. Engrs.*, Part 2, March, pp 1-18
- ALI, K.H.M., and WALLEY, P., (1992). "Flow fields in scour holes generated by offset jets at different heights above the original bed level", *Internal Report, Dept. of Civil Engineering, The University of Liverpool*
- ALTINBILEK, H.D. and OYKAY, S. (1973). "Localised scour in a horizontal sand bed under vertical jets." *Proc. IAHR Congress*, International Association for Hydraulic Research, 1, pp 99-106
- BREUSERS, H.N.C., (1967). "Time scale of two-dimensional local scour." *Proc. 12th Congress, IAHR*, Fort Collins, Vol. 3. pp C321-8
- BREUSERS, H.N.C., (1975). "Computation of velocity profiles in scour holes" *Proc. 16th Congress, IAHR*, Sao Paulo, Vol. 2. pp C300-306
- DODDIAH, D., ALBERTSON, M.L. and THOMAS, R. (1953). "Scour from jets." *Proc. IAHR Congress*, International Association for Hydraulic Research, pp 161-169
- FLUENT USER'S Guide, (1993)., "Fluent Incorporated, Lebanon, New Hampshire, U.S.A.
- FRANCIS, J.R.D. and GHOSH, S.N. (1974). "A new look at local erosions in alluvial rivers." *Proc. 5th Australasian Conf on Hydr. and Fluid Mech.*, University of Canterbury, Canterbury, New Zealand, pp. 71-77
- HASSAN, N.M.K. and NARAYANAN, R. (1985). "Local scour downstream of an apron." *J. Hydr. Engrg., ASCE*, 111(11), pp 1371-1385
- IWAGAKI, Y., SMITH, G.L. and ALBERTSON, M.L. (1958). "Analytical study of mechanics of scour for three dimensional jet." *Hydr. Conf., ASCE*, New York, N.Y.
- JOHNSTON, A.J. (1990). "Scourhole developments in shallow tailwater." *J. Hydr. Res., IAHR*, 28(3), pp 341-354
- KARIM, O.A. (1996)., "Prediction of two and three dimensional turbulent flows", *Ph.D. Thesis, University of Liverpool*
- LIM, S.Y., (1985), "Scour and particle diffusion caused by water jets", *Ph.D. Thesis, University of Liverpool, U.K.*
- MASON, P.J. (1988). "Effects of air-entrainment on plunge pool scour." *J. Hydr. Engrg., ASCE*, 115(3), pp 385-399
- MELVILLE, B.W. and RAUDKIVI, A.J. (1977). "Flow characteristics in local scour at bridge piers." *J. Hydr. Res.*, Vol. 15, No. 4, pp 373-380
- POREH, M. and HEFEZ, E. (1967). "Initial scour and sediment motion due to an impinging jet." *Proc. IAHR Congress*, International Association for Hydraulic Research, 3, 8
- RAJARATNAM, N., (1976). "Turbulent jets." Elsevier Scientific Publishing Co., Amsterdam, The Netherlands, 1976
- RAJARATNAM, N. and BELTAOS, S., (1977). "Erosion by impinging circular turbulent jets". *J. Hydr. Div., ASCE*, 103(10), pp 1191-1205
- RAJARATNAM, N. and BERRY, B., (1977). "Erosion by circular turbulent wall-jets". *J. Hydr. Res.*, 15(3), pp 277-289
- RAJARATNAM, N. and HUMPHRIES, J.A., (1983). "Diffusion of Bluff Wall Jets in Finite Depth Tailwater". *J. of Hydr. Engineering, ASCE*, Vol. 109, No. 11, pp 1471-1486
- RAJARATNAM, N. and MACDOUGALL, R.K.B., (1983). "Erosion by plane wall jets with minimum tailwater." *J. Hydr. Div., ASCE*, 109(7), pp 1061-1064
- ROSIŃSKI, K.I., (1961). "Hydraulic of scouring pits". *Chapter 2 from "Problems of River Runoff Control", Acad. Sci. USSR*, Section for Scientific Study of Water Engineering Problems
- ROUSE, H., (1939). "Criteria for similarity in the transportation of sediment". *Proc. Hyd. Conf. Studies Engineering Bull.*, Univ. of Iowa, pp 33-49
- SALEHI-NEYSHABOURY, A.A. and ALI, K.H.M., (1990)., "Flow field of an offset jet", *International Conference on Physical Modelling of Transport and Dispersion*, Massachusetts, pp 10A.13-10A.18
- SARMA, K.V.M. and SIVASANKAR, R. (1967). "Scour under vertical circular jets". *J. Inst. of Engrs.*, Calcutta, India 48(3), pp 568-579
- UYUMAZ, A., (1988). "Scour downstream of vertical gate". *J. Hydr. Engrg., ASCE*, 114(7), pp 811-816
- WESTRICH, B. and KOBUS, H., (1973). "Erosion of a uniform sand bed by continuous and pulsating jets". *Proc. IAHR Congress*, International Association for Hydraulic Research, 1, pp 91-98
- WU, S. and RAJARATNAM, N., (1995). "Free jumps, submerged jumps and wall jets". *J. of Hydr. Res., ASCE*, Vol. 33, NO. 2. pp 197-212.

