Building Systems: Room Air and Air Contaminant Distribution. University of Illinois 5-8 Dec. 1988

3868

NUMERICAL SOLUTION OF WALL JETS

H B Awbi and A A Setrak
Napier Polytechnic, Edinburgh, UK

ABSTRACT

This paper describes a numerical solution procedure which uses the finite volume method and the k -ε turbulence model for predicting the flow of isothermal and non-isothermal wall jets. This was used to predict the velocity and temperature profiles of the jets. The procedure is also extended to predict the flow of two and three-dimensional wall jets over a surface-mounted obstacle. The numerical predictions are corroborated by experimental results.

1. INTRODUCTION

Most air jets used in room ventilation are supplied over a surface or ultimately become attached to a surface due to the Coanda effect. An air jet that is bound by a flat surface on one side is normally known as a "wall jet". After discharge from the supply opening, the initial flow of a jet (usually referred to as the primary air) has a major influence on the air motion in the room. To achieve efficient mixing of the jet with room air it is necessary to know not only the initial condition of the air supply and its position in the room but also its aerodynamic characteristics up to the throw length (ie. the distance from the supply where the maximum jet velocity decays to a specified value such as 0.25 or 0.5 m/s). For the purpose of air distribution design, the characteristics of isothermal jets may be obtained from air diffusion standards and handbooks (1), (2) and (3) or from nomograms provided by air diffusion equipment suppliers. In most instances such information is only available for isothermal wall jets over a smooth surface. However, information on the diffusion of non-isothermal wall jets and wall jets affected by physical barriers or by other jets is, until recently, obtained from experimental measurements in special test facilities. Because of the low velocity and high turbulence associated with ventilation jets such measurements are often difficult to carry out accurately and at best do not provide sufficient detail of the flow.

As a result of recent advances in computational fluid dynamics it is now possible to predict, with reasonable accuracy, the diffusion of air jets and the air movement in ventilated rooms. Awbi and Setrak (4) and (5) have developed a numerical procedure using the finite volume method to predict the diffusion of isothermal and non-isothermal plane (two-dimensional) wall jets. The effect of surface-mounted rectangular obstacles and surface roughness elements on the diffusion of a two-dimensional wall jet was also well predicted. In this paper, the finite volume solution procedure has been extended to predict the flow of three-dimensional wall jets. The predicted results are compared with experimental data obtained from an air jet rig as well as published experimental data. The flow of two and three-dimensional air jets over a surface-mounted obstacle was also predicted using the same numerical procedure and the results are compared with experimental measurements.

2. NUMERICAL SOLUTION

The steady, incompressible and turbulent flow of a jet, in which the fluctuating velocities and heat fluxes are described by a turbulence model may be represented by the general equation:

$$\frac{\partial}{\partial x_i}(\rho U_i \phi) = \frac{\partial}{\partial x_i}\left(\Gamma_\phi \frac{\partial \phi}{\partial x_i}\right) + S_\phi \qquad (1)$$

where U_i is the mean velocity component in the direction x_i, ϕ is the dependent variable, Γ_ϕ is the diffusion coefficient for ϕ, and S_ϕ represents the source terms for ϕ. In the equations for continuity, momentum (Navier-Stokes) and thermal energy, the variable ϕ acquires respectively the values 1, U_j (ie. mean velocity in direction x_j) and the mean temperature T. The turbulence model used in this investigation is a two-equation model representing the kinetic energy, k, and its rate of dissipation, ε (ie. k - ε turbulence model). Therefore, two additional transport equations with k and ε as the dependent variables need to be solved. These two equations include in their source terms expressions for the effect of buoyancy on the turbulence energy which are used in the solution of non-isothermal jets. Detailed description of the transport equations and the turbulence model used is given by Awbi (6).

Using a rectangular finite difference grid for the flow domain, equation (1) may be discretised by integration (summation) over an element in a control volume or cell to yield an algabraic equation of the form:

$$(\Sigma a_i - S_p)\phi_p = \Sigma(a_i \phi_i) + S_u \qquad (2)$$

which relates nodal values of the variable at node p, ϕ_p, to values at six neighbouring nodes, f_i, (ie. N,S,E,W,R,L). The coefficient a_i in the finite difference equation (2) links the convective and diffusive terms of the differential equation between cell P and cell i. In equation (2) S_u and S_p are the source and sink terms of cell p and the term (S_u + S_p ϕ_p) is integrated over the cell. This method of solution is known as the SIMPLE algorithm which is described by Patankar (7) and used by many investigators. The method does not employ equations for pressure, instead the pressure in each cell is "linked" to the velocities of the surrounding cells so that continuity is always observed. This solution procedure requires the use of a staggered grid where the scalar quantities (P,T,k,ε) are located at a grind point P and the velocity components U,V and W are located at intermediate locations between grid points. In this investigation the interest lies in the flow close to the wall and around surface-mounted obstacles hence a non-uniform grid was chosen for x-and-y directions (axial and vertical respectively) and a uniform grid was used for the z-direction (lateral). A grid size of 41 x 27 was used for two-dimensional flows and 20 x 20 x 11 for three-dimensional flows.

To obtain a solution to the transport equations boundary conditions need to be specified. In this case, the boundary conditions are imposed at the jet inlet and exit, at the walls and the surfaces of obstructions and at the free (entrainment) boundary. Across the jet inlet a uniform distribution of U,T,k and ε is assumed. The exist velocity and temperature is calculated using the continuity and energy balance equations respectively. At the walls and obstacle surfaces wall function expressions are used for the shear stresses and heat fluxes. At the free boundary of the jet the longitudinal velocity is taken zero and the vertical entrainment velocity is specified as:

$$V = C_e U_m \quad (3)$$

where U_m is the maximum longitudinal velocity in a vertical plane across the jet and C_e is an entrainment coefficient taken as 0.035 for a two-dimensional wall jet and 0.03 for a three-dimensional wall jet (8). The temperature of the free boundary is also used as a boundary condition for the energy equation.

Further details of the numerical solution procedure are found in references (6) and (8).

3. RESULTS

3.1 Wall Jet Flows

The velocity and temperature profiles for a wall jet were predicted for different values of Reynolds number, Re, and Archimedes number, Ar, and the results were compared with experimental data. For a wall jet these non-dimensional parameters are defined as:

$$Re = U_o d/\upsilon, \qquad Ar = \beta g d \theta_o / U_o^2$$

where U_o is the inlet velocity, θ_o is the difference between the supply temperature and room temperature, β is the coefficient of thermal expansion and d is the size of the supply opening which is taken as the slot height, h, for two-dimensional jets or $\sqrt{A}$ for three-dimensional jets where A is the effective area of the opening. For a range of Re between 1.2×10^3 to 3.5×10^4 the normalised velocity profiles in the fully developed region of the jet were found to be indepdent of Re. Figures 1 and 2 show the predicted velocity and temperature profiles in the fully developed region of a two-dimensional wall jet compared with experimental data. Here, U_r/U_m is the ratio of the resultant velocity, U_r, at a point to the maximum jet velocity, U_m, in a plane containing the point and $y/y_{0.5}$ is the ratio of the distance of a point from the surface to the distance of the point where U = 0.5 U_m. The ratio θ/θ_m is the difference in temperature between a point in the jet and the supply temperature, θ, to the difference between the maximum (or minimum) temperature across the jet and the supply temperature, q_m. These plots show no definite effect of Ar on the resultant velocity profiles and the agreement with experimental data is good. The lower velocities due to Albright and Scott (9) is attributed to the fact that only the U-component of velocity has been used in their profiles. The decay of the maximum velocity in the axial direction is shown in Fig. 3 for a rectangular opening of aspect ratio 3.3. The predicted results are compared favourably with the measured data and the experimental results of Rajaratnam (10).

3.2 Wall Jets with Obstacles

When an obstacle is placed in the path of a wall jet (such as a beam, a light fitting, furniture etc) the jet either separates from the surface completely or reattaches downstream of the obstacle. The course which the jet takes will depend on the distance of the obstacle from the supply opening, the geometry and size of the obstacle, the size of the supply opening, the Archimedes number etc. Setrak (8) has shown that complete separation is more likely to occur in the case of an infinitely long obstacle placed in the path of a two-dimensional wall jet when the distance of the obstacle from the supply is smaller than a minimum distance known as the "critical distance". Complete separation is unlikely to take place when a long obstacle is placed in the path of a three-dimensional wall jet or a finite length obstacle is placed in the path of a two-dimensional jet unless the Archimedes number exceeds 0.01.

Figure 4 shows normalised velocity profiles for a three-dimensional wall jet in the presence of an infinitely long obstacle placed on the wall. The jet reattachment downstream of the obstacle produces a thicker boundary layer than would otherwise exit at the same position in the absence of the obstacle. The velocity profiles for a two-dimensional jet which separates due to the presence of a two-dimensional obstacle are shown in Fig. 5. The velocity profile downstream of the obstacle resembles that for a free jet where entrainment of the surrounding air now occurs at two free boundaries and the jet decays faster than a reattached jet. The predicted profiles in Figs. 4 and 5 are seen to be corroborated by the measured profiles using hot wire and hot film anemometers. Figure 6 shows predicted velocity vector and streamline plots for a two-dimensional wall jet separating from the surface due to the presence of a two-dimensional obstacle. The entrainment vortex can clearly be seen between the supply opening and the obstacle.

4. CONCLUSIONS

The numerical solution described in this paper is found to predict the diffusion of two-and three-dimensional wall jets with sufficient accuracy for most ventilation applications. This method has also accurately predicted the complex flows involving a wall jet and a surface-mounted obstacle where the jet either completely separates or reattaches to the surface. The same procedure may be applied to predict the air movement in enclosures and the dispersion of contaminants in a room.

REFERENCES

1. ISO 5219: Air distribution and air diffusion - Laboratory aerodynamic testing and rating of air terminal devices, 1984
2. BS 4773: Pt 1, Methods for testing and rating of air terminal devices for air distribution systems - aerodynamic testing, 1971.
3. ASHRAE Fundamentals Handbook, Space air diffusion, chapter 32, 1985.
4. Awbi, H B and Setrak, A A, Numerical solution of ventilation air jet, Proc. 5th Int. Symp on the Use of Computers for Environmental Engineering Related to Buildings, Bath, UK, July 1986, pp 236-246.
5. Awbi, H B and Setrak A A, Air jet interference due to ceiling-mounted obstacles, Air Distribution in Ventilated Spaces Conference, Stockholm, June 1987.
6. Awbi, H B, Application of computational fluid dynamics in room ventilation, Building and Environment, Vol. 24, No. 1, 1989.
7. Patankar, S V, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, 1980.
8. Setrak, A A, The Effect of Rectangular Obstacles on the Diffusion of a Wall Jet, PhD thesis, Napier Polytechnic, UK, Nov. 1988.
9. Albright, L D and Scott, N R, The low-speed non-isothermal wall jet, J Agric. Eng. Res, Vol. 19, pp 25 - 34, 1974.
10. Rajaratnam, N and Pani, B S, Three-dimensional turbulent wall jets, J Hyd. Division, ASCE, Vol. 100, pp 69 - 83, 1974.

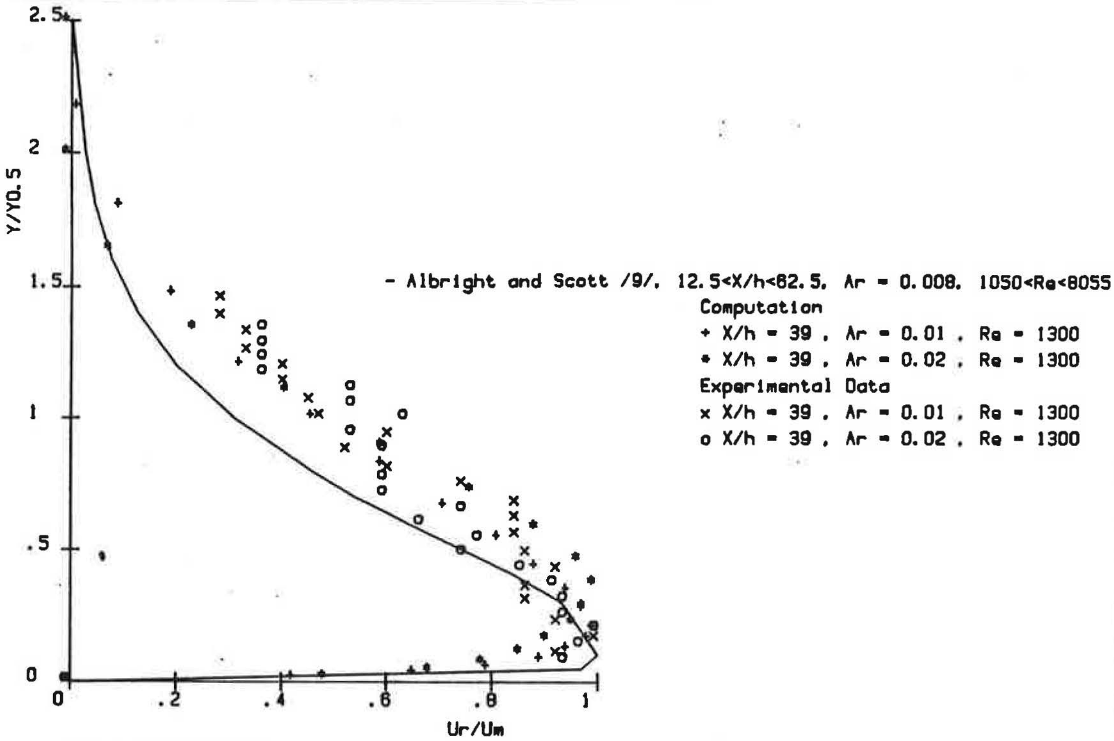

Fig 1 Experimental and Computed velocity profiles for a non-isothermal wall jet

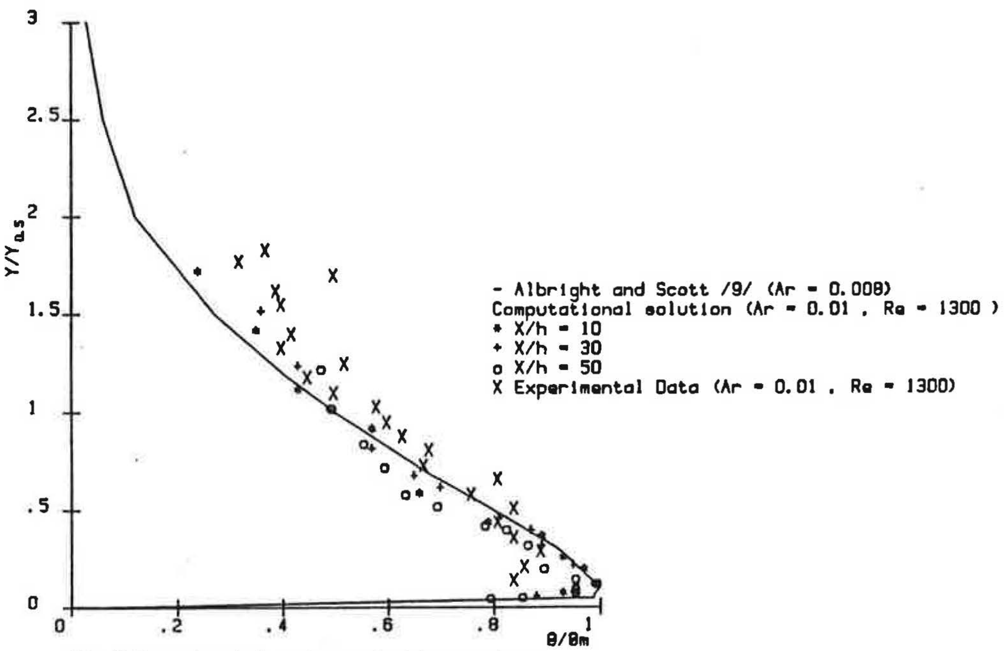

Fig 2 Experimental and computed temperature profiles for a plane wall jet

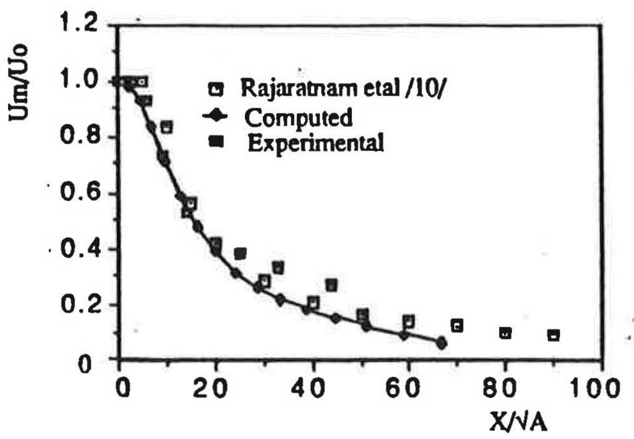

Fig 3 Decay of maximum velocity of a three-dimensional wall jet.

$$Re = 4.3 \times 10^4,\ \frac{b}{h} = 53.6,\ \frac{x_d}{\sqrt{A}} = 9.4,\ \frac{d}{\sqrt{A}} = 0.78$$

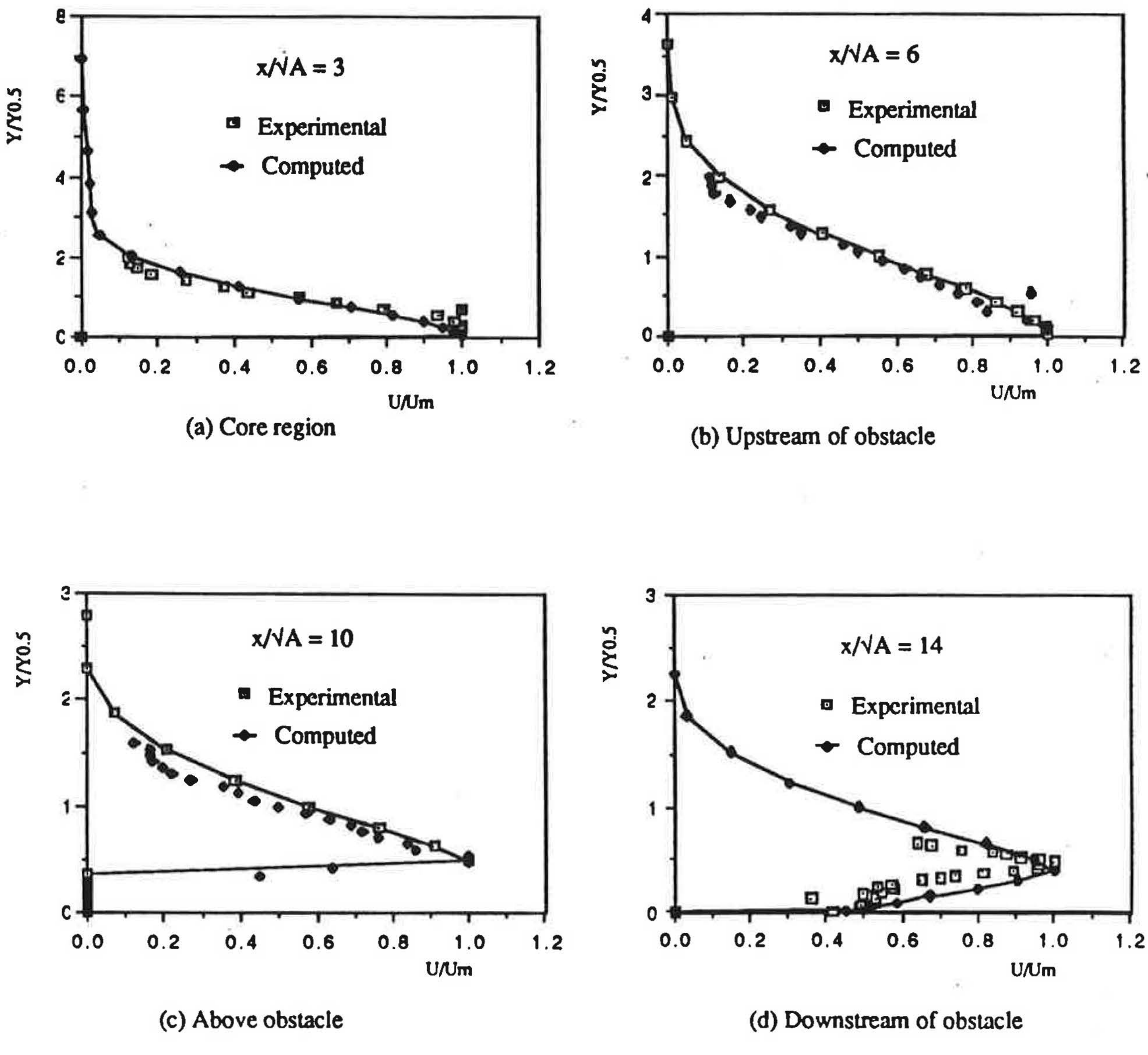

Fig 4 Experimental and computed velocity profiles for a rectangular inlet with a long obstacle.

d/b = 1.6, d/h = 2.68, Xd/h = 31, Re = 3.5×10^4

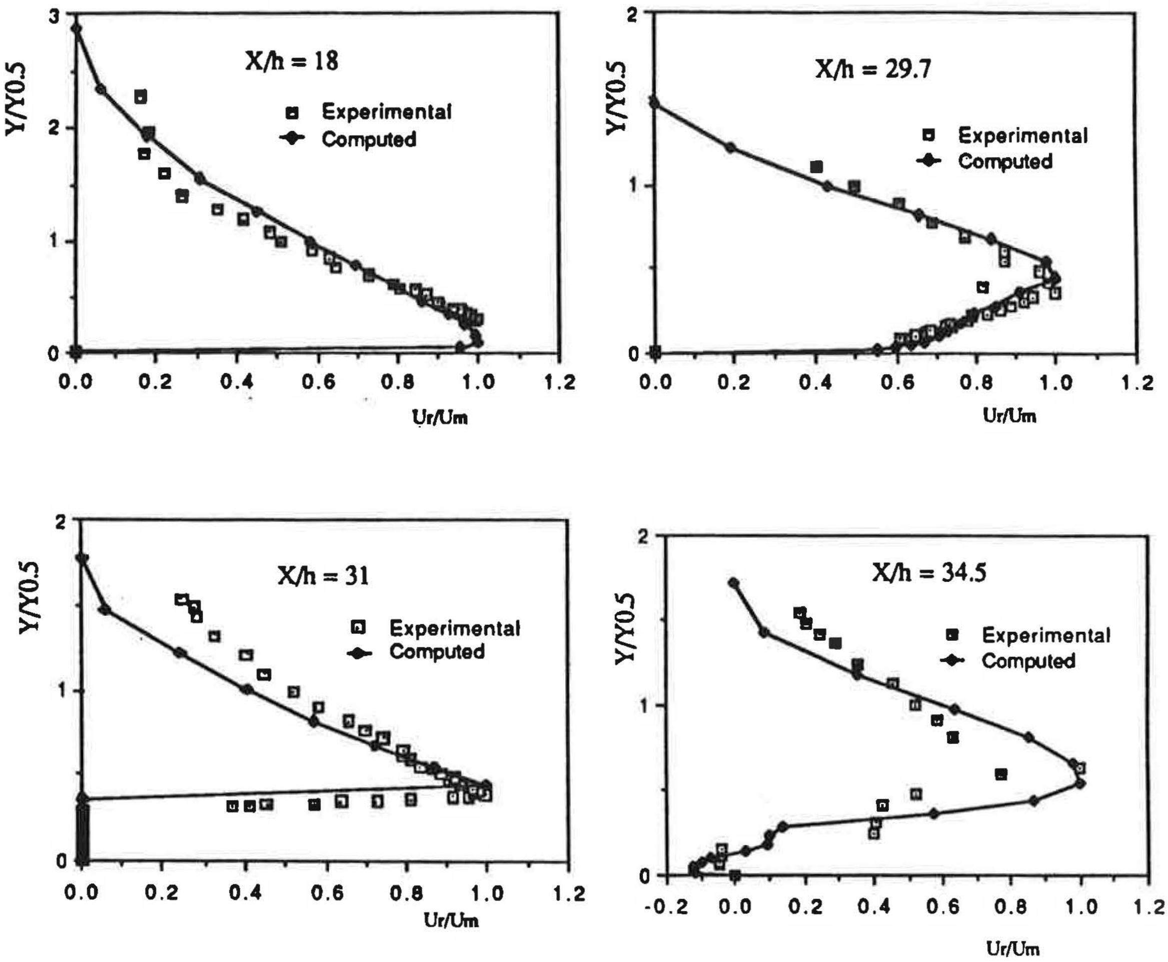

Fig 5 Experimental and computed velocity profiles for separated wall jet due to an obstacle.