Roomvent-87

#3856

AIR JET INTERFERENCE DUE TO CEILING-MOUNTED OBSTACLES

H B Awbi and A A Setrak
Department of Mechanical and Industrial Engineering,
Napier College, Edinburgh, U K

SUMMARY

The effect of surface-mounted single and multiple obstacles in the path of a two-dimensional wall jet is investigated experimentally and theoretically using a finite difference solution of the flow conservation equations. It is found that both the height, measured from the surface, and depth along the flow of an obstacle affect the distance from the supply slot at which the jet is about to separate, ie. the critical distance. The presence of an obstacle accelerates the decay of the jet and the decay increases further when the jet separates from the surface. With multiple obstacles the diffusion of the jet increases as the relative height of the obstacles d/h increases.

1. INTRODUCTION

From a thermal comfort viewpoint it is required to know the air velocity and temperature distribution in the occupied zone of a room. However, from energy utilisation considerations it is also required to know the velocity and temperature distribution outside, as well as within, the occupied zone. Temperature stratification and high air velocities close to room surfaces influence the energy demand for the air conditioning plant. The flow both inside and outside the occupied zone is determined by the initial conditions of the air jet as well as the manner in which the air jet is diffused in the conditioned room and whether the jet is influenced by physical barriers or obstructions along its flow path (1,2).

The air distribution system designer relies on diffuser performance data supplied by the manufacturer. This data is usually obtained by testing the diffuser in accordance with some standard prevailing in the manufacturer's country (3) or the ISO standard (4). These standards specify minimum room dimensions, experimental set-up, instrumentation type and accuracy as well as a recommended test procedure for different devices. The standards assume isothermal conditions as the basis for evaluating the aerodynamic characteristics of grilles and diffusers and in the case of a surface-bound air supply, a smooth surface only is considered. These assumptions are seldom realised in pratice particularly with buoyant flows and when obstructions such as structural beams and light fittings are present in the path of a ceiling jet. Such factors could significantly affect the air velocity and temperature distribution in the occupied zone (5-9).

Until recently, these effects could only be evaluated by physical modelling which is expensive, time consuming and the results could be influenced by scaling effects (10). With recent advances in mathematical modelling techniques, a number of investigators have developed computer programs for solving flow problems involving large recirculating zones (11-13). These programs have been adapted by numereous workers to solve various ventilation problems ranging from clean room applications (14), general room air movement studies (15), air jet diffusion (2), to predicting fire spread in a building (16).

In this paper the factors which influence the diffusion of a wall jet that are not treated by diffuser testing standards are studied experimentally and theoretically. The effects of buoyancy and both single and multiple obstacles in the path of a two-dimensional wall jet are considered. A wall jet test facility was employed for the experimental study and a finite difference computer program was used for the theoretical solution.

2. NUMERICAL SOLUTION

The air velocity and temperature distributions in a ventilated room are usually calculated by solving, in finite difference form, the conservation equations of momentum, energy and mass which govern the air flow and convective heat transfer in the room either in two or three-dimensions. The effect of turbulence is described by means of a suitable turbulence model which represents the stochastic processes of turbulence by steady-state equivalents. A widely used turbulence model is a two-equation model representing the kinetic energy of turbulence, k, and its dissipation rate, ε. This is usually referred to as the $k-\varepsilon$ turbulence model and has been used by many investigators. When this model is applied the two conservation equations of k and ε are also solved in finite difference form in addition to those equations of momentum, energy and mass.

The computer program TEACH (12), which was written to solve the two-dimensional conservation equations, has been adapted for solving the flow of a wall jet. Because in this instance the interest lies in the flow close to the wall and around wall-mounted obstacles a non-uniform finite difference grid was chosen to allow for a very fine grid in those regions for improved accuracy. Momentum and energy entrainments at the free boundary of the jet were incorporated in the boundary conditions. To predict separating and reattaching flows due to the presence of obstacles in the path of the jet a subroutine was written for this purpose which specifies the boundary conditions around the obstacles. For a non-isothermal flow buoyancy terms were added to the conservation equations of the vertical component of velocity, the kinetic energy and its dissipation rate to allow for the buoyancy force due to the temperature difference within a computational cell. Further details of this program are found in references (12) and (2).

3. EXPERIMENTAL FACILITY

A wall jet rig comprising a fan, an electric heater, a damper, a plenum chamber, a variable nozzle and a channel open at the top and at one end was used for the experimental investigation, Fig. 1. The air jet leaves the nozzle to enter a channel 5m long and 1m wide were measurements are made. The nozzle which had the same width as the channel was of variable height ranging from 19 to 35mm.

The velocity measurements were carried out using three different types of DANTEC constant temperature hot wire or hot film anemometers according to the flow situation being investigated. An x-array hot wire probe was used for measuring the two velocity components in isothermal flows involving velocities greater than about 2m/s, a temperature compensated hot wire probe was used for measuring velocities in excess of 2m/s in non-isothermal flows and an omini-directional temperature-compensated hot film was used for measuring low velocities in both isothermal and non-isothermal flows. The temperatures were measured with screened thermocouple wire. The Reynolds number for the tests with isothermal flow was 19000 and that for non-isothermal flow was 1900

The velocity distribution across the nozzle showed a flat profile except very close to the side walls of the channel. However, all measurements were carried out in the centre of the channel to ensure the presence of two-dimensional flow.

The movement of the instruments across the jet was accurately performed using a stepper motor under computer control. The accuracy in positioning the anemometers across the jet is estimated to be ±0.1 mm for the hot wires and ± 0.5mm for the hot film. The collation and analysis of data was also carried out using the same personal computer. The accuracy in the velocity measurements is estimated to be ± 4%.

4. RESULTS AND DISCUSSION

4.1 Plane Wall Jet Without Obstruction

The plane wall jet has been extensively investigated and experimental data for both isothermal and non-isothermal jets is found in the literature (17, 18).

To validate the numerical solution, normalised velocity and temperature profiles in the fully developed region of the jet (ie. downstream of the core) obtained using the numerical solution are compared with experimental results in Fig. 2 (a and b). It is shown that the theoretical profiles are very close to the experimental ones.

Figure 3 shows the decay of the maximum velocity of the jet U_m with distance from the nozzle. The theoretical predictions which are also substantiated by experimental data for isothermal flow and non-isothermal flow at $Ar = 0.012$ show that the effect of buoyancy on a hot jet moving over a floor or a cold jet on a ceiling is to increase the diffusion process and this subsequently decreases the maximum jet velocity more rapidly. For Archimedes numbers Ar greater than about 0.02 the buoyancy force overcomes the Coanda effect and the jet separates from the surface. Similar observations were made by Nielsen et al (15) for a wall jet in a room. In normal ventilation applications this is a very undesirable situation as complete mixing of the jet with room air cannot be attained and furthermore large velocities will be present in the region where the jet penetrates the occupied zone.

4.2 Attached Wall Jet with a Single Obstacle

When a wall jet encounters an obstacle attached to the wall, Holmes and Sachariewicz (19) found that the jet can take one of three courses:

(i) almost unaffected by the obstacle;
(ii) separates from the surface and reattaches downstream of the obstacle;
(iii) completely separates from the surface.

There are a number of factors that can influence the jet behaviour, such as the size of the obstacle, its distance from the inlet, the Reynolds number, the turbulence level of the supply, the temperature difference between the supply and the room etc. Holmes and Sachariewicz studied the effect of the height of a square section obstacle and its distance from the inlet on the flow in a purpose built wall jet test rig. Nielsen (8, 9) studied experimentally the effect of obstacle height, its distance from the inlet and the Archimedes number on the penetration depth of the jet into the occupied zone of a room. However, none of these investigations studied the effect of the height to depth ratio d/b of the obstacle. Previous work with rectangular prisms in a wind tunnel by one of the authors (20) showed a significant influence of d/b on the flow around the prisms. In room air conditioning practice, large variations in d/b can be present in integrated ceiling designs incrorporating light fittings and beams.

Figure 4 shows the measured and predicted velocity profiles at a distance of 0.26 h downstream of a single obstacle of $d/b = 0.8$ and $x_d/h = 31$ representing a reattached flow similar to type (ii) mentioned earlier. Knowing the steep velocity gradients in the vicinity of a flow separation the numerical solution predicts the velocity distribution satisfactorily. The decay of the maximum velocity of the jet is shown in Fig. 5 and here again good agreement is found between the numerical solution and the experimental measurements. The figure shows that the maximum jet velocity decreases as the flow approaches the obstacle, then abruptly increases as the jet passes over the obstacle, and finally gradually decreases after reattachement. In general, the obstacle causes a faster decay of the jet velocity as a result of the increased diffusion due to localised separation in the vicinity of the obstacle. A theoretical flow pattern for a reattached flow over an obstacle is shown as a velocity vector plot in Fig. 6.

4.3 Separated Wall Jet With a Single Obstacle

Figure 7 shows the velocity profile across a separated wall jet at a distance of 0.26h downstream of an obstacle. The position of the obstacle is the same as that for Fig. 4, however, the height of the obstacle was increased to cause the jet to separate. In Fig. 4, $d/h = 1.3$ and in Fig. 7 $d/h = 2.68$. By comparing these two figures it is seen that a greater diffusion of the shear layer (outer boundary of jet) occurs downstream of the obstacle when the jet reattaches to the surface. However, by comparing the decay of the maximum velocity for the separated jet, Fig. 8, with that for the reattached jet, Fig. 5 it is clear that the velocity decay for the separated jet is faster since after separation, the jet behaves as a free plane jet entraining air on both sides. With a separating jet it was only possible to obtain velocity traverses close to the obstacle as the jet leaves the channel shortly after separation. The theoretical flow pattern for a wall jet separating over an obstacle is depicted in Fig. 9.

The effect of the obstacle dimensions on the critical distance x_c (ie. the minimum distance from the supply slot at which the jet reattaches to the surface downstream of the obstacle) is depicted in Fig. 10. These results are for an isothermal jet and the critical distance was determined by smoke visualisation. It is clear that the height to depth ratio d/b of the obstacle has a major influence on the critical distance. An increase in b/h from 1.2 to 1.6 almost doubles x_c for the same value of d/b.

The theoretical prediction of the critical distance for an obstacle of $b/h = 1.65$, although underestimates the effect of the depth of the obstacle, is in general agreement with the experimental results. It should be noted that with using smoke it is difficult to determine the precise position of the critical point. This is mainly due to the flow close to the obstacle being highly disturbed causing rapid dispersion of the smoke and furthermore the flow exhibits a hysteresis effect, ie. different values of x_c can be obtained when the obstacle is moving away from the supply and when it is approaching it (19). Results obtained from reference (19) are also plotted in Fig. 10 for square obstacles moving away from the slot. Considering the difficulty in determining the critical distance and the differences in the experimental conditions, the agreement with the present results is satisfactory.

4.4 Wall Jet with Multiple Obstacles

In practice, the ceilings of ventilated rooms are not always smooth and often roughness elements are present either for the purpose of lighting distribution or for an aesthetic purpose or both. The effect of uniform ceiling roughness on the diffusion of wall jet is not treated in diffuser testing standards nor is it usually considered in diffuser nomograms. Surface roughness can substantially accelerate the decay of maximum jet velocity as shown in Fig. 11. Consequently, the throw of the jet can be greatly reduced resulting in a deficient diffusion of the jet with room air. The figure shows a comparison between the maximum velocity decay for a jet over rough surfaces with that for a smooth surface, both flows being isothermal. Experimental results reported by Rajaratnam (17)for roughness ratio d/h up to 0.12 are also plotted and show a close agreement with the results from the numerical solution for roughness heights of similar values. As the height of the roughness blocks increase the decay of the maximum velocity becomes more rapid due to the increased diffusion of the jet with the surrounding stagnant air caused by localised separation at each obstacle. The predicted results are for a pitch ratio P/h of 6.6. However, in one case P/h of 5.1 was also used without any significant effect on the velocity decay.

5. CONCLUSIONS

The presence of an obstacle in the path of a wall jet enhances the diffusion of the jet and accelerates the decay of maximum velocity even when the jet remains attached to the surface. The throw of the jet decreases as a result and this ought to be considered in the design of air distribution in rooms with ceiling-mounted or wall-mounted obstacles.

It was found that both the height d and depth b of an obstacle relative to the height of the supply slot h influence the critical distance of the obstacle from the slot x_c . The critical distance increases as the obstacle becomes more 'streamlined', ie. as b/h increases. for the same aspect ratio.

When the jet separates, the maximum velocity decays faster than when it reattaches to the surface downstream of an obstacle. However, where the separating jet enters the occupied zone a large increase in the room velocity is expected in that region which will affect the thermal comfort in the room.

For a jet flowing over a rough ceiling, the decay of the jet velocity increases as the height of the roughness elements increases. No significant effect of the pitch of the elements on the jet diffusion was detected.

The finite difference solution produced results which were in general agreement with experimental data for a number of complex flows involving extensive recirculating zones. This demonstrates the potential of numerical solutions in predicting room air movement particularly at the design stage.

NOMENCLATURE

Ar	Archimedes number $(\beta g h \theta_o / U_o^2)$
b	width of obstacle
d	height of obstacle
g	gravitational acceleration
h	height of supply slot
p	obstacle pitch
Re	Reynolds number $(U_o h / \nu)$
U_m	maximum jet velocity
U_o	slot velocity
x	axial distance from slot
x_c	critical axial distance of obstacle from slot
x_d	axial distance of obstacle from slot
y	distance normal to surface
$y_{0.5}$	distance normal to surface where $u = \frac{1}{2} U_m$
β	coefficient of thermal expansion
θ	difference in temperature between the slot and jet
θ_m	difference between the slot air temperature and the maximum jet temperature
θ_o	difference in temperature between the slot and room air
ν	kinematic viscosity of air

REFERENCES

1. Whittle, G E, Room air distribution: design and evaluation, BSRIA Tech. Note 4/86, Bracknell, 1986.

2. Awbi, H B and Setrak, A A, Numerical solution of ventilation air jet, Proc. 5th Int. Symposium on the Use of Computers for Environmental Engineering Related to Buildings, Bath, England, July 1986, pp 236 - 246,

3. BS 4773 Part 1, Methods for testing and rating air terminal devices for air distribution systems - Aerodynamic testing, BSI, 1971.

4. ISO Standard 5219, Air distribution and air diffusion - Laboratory aerodynamic testing and rating of air terminal devices, 1984.

5. Awbi, H B and Savin S J, Air distribution methods for domestic warm air heating systems using low grade heat sources - tests with low level and high level air supply terminals, Commission of the European Communities Report EUR 9237 EN, 1984.

6. Moog, W and Sodec, F, Raumstromungsuntersuchungen fur das Projekt Stadthalle Aachen, Heizung Luftung Haustechnik, Vol. 27, pt 12, pp 442 - 448, 1976.

7. Sollner, G and Klinkenberg, K, Leuchten als storkorper im luftstrom, Heizung Luftung Haustechnik, Vol. 23, pt 4, pp 113 - 116, 1972.

8. Neilsen, P V, The influence of ceiling-mounted obstacles on the air flow pattern in air-conditioned rooms at different heat loads, Building Services Engineering Research and Technology, Vol. 1, No. 4, pp 199 - 203, 1980.

9. Nielsen, P V, Air diffusion in rooms with ceiling-mounted obstacles and two-dimensional isothermal flow, Proceedings of the 16th International Congress of Refrigeration, Paris 1983, pp 161 - 168.

10 Rolloos, M, Possibilities and limitations for the prediction of air flow patterns, temperature and velocities in large halls using scale models, Meeting of Commission E1 of the International Institute of Refrigeration, Belgrade, Yugoslavia, pp 245 - 256, 1977.

11 Moult, A and Dean, R B, CAFE - a computer program to calculate the flow environment, CAD 80, 4th International Conference on Computers in Design Engineering, Brighton, 1980.

12 Gosman, A D and Ideriah, F J K, TEACH-2E: A general computer program for two-dimensional, turbulent, recirculating flows, Department of Mechanical Engineering, Imperial College, London 1983.

13 PHOENICS Code, Concentrated Heat and Momentum Ltd, Wembley, London, 1986.

14 Broyd, T W, Deaves, D M and Oldfield, S G, The use of computational modelling techniques for cleanroom design,Proceedings of the Semi conductor Processing and Equipment Symposium, Zurich, Switzerland 1983.

15 Nielsen, P V, Restivo, A and Whitelaw, J H, Buoyancy-affected flows in ventilated rooms, Numerical Heat Transfer, vol. 2 pp 115 - 127, 1979.

16 Cox, G, Computational fire dynamics, Chartered Mechanical Engineer, Vol 33, No 718, pp 29 - 31, 1986.

17 Rajaratnam, N, Turbulent Jets, Elsevier, 1976.

18 Albright, L D and Scot, N R, The low-speed non-isothermal wall jet, J Agric Engng Res, Vol 19, p 25-34, 1974.

19 Holmes, M J and Sacharievicz, E, The effect of ceiling beams and light fittings on ventilating jets, HVRA Laboratory Report No 79, BSRIA, Bracknell, 1973.

20 Awbi, H B, Wind-tunnel wall constraint on two-dimensional rectangular-section prisms, Journal of Industrial Aerodynamics, vol 3, pp 285 - 306, 1978.

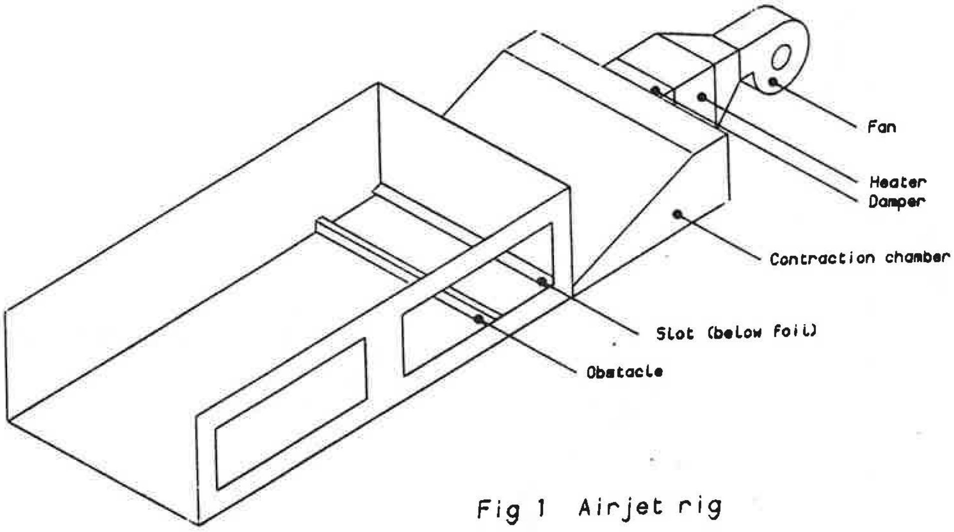


Fig 1 Airjet rig

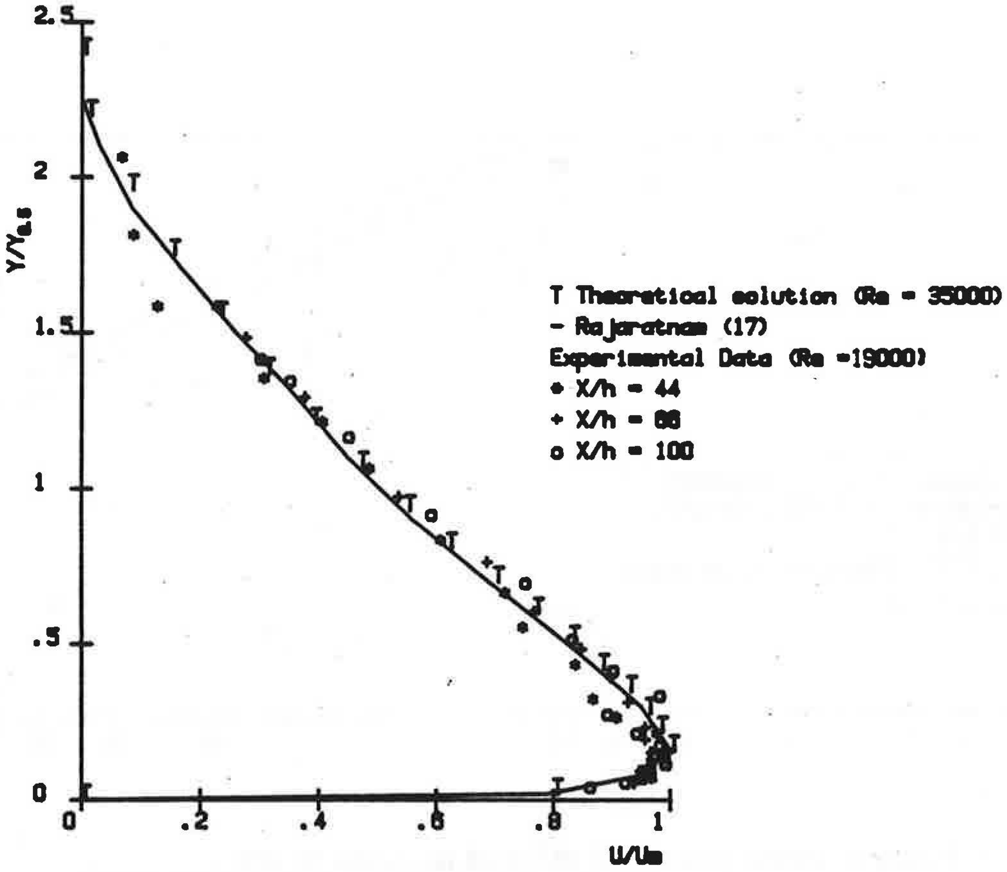


Fig 2a Experimental and theoretical velocity profiles for a plane wall jet

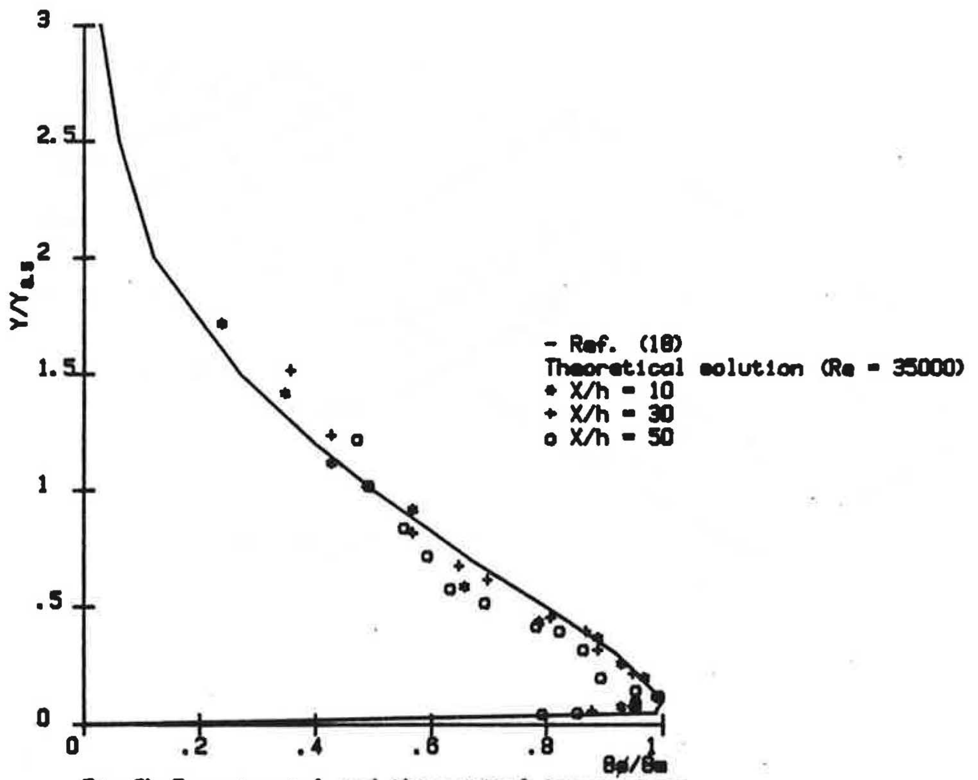


Fig 2b Experimental and theoretical temperature profiles for a plane wall jet

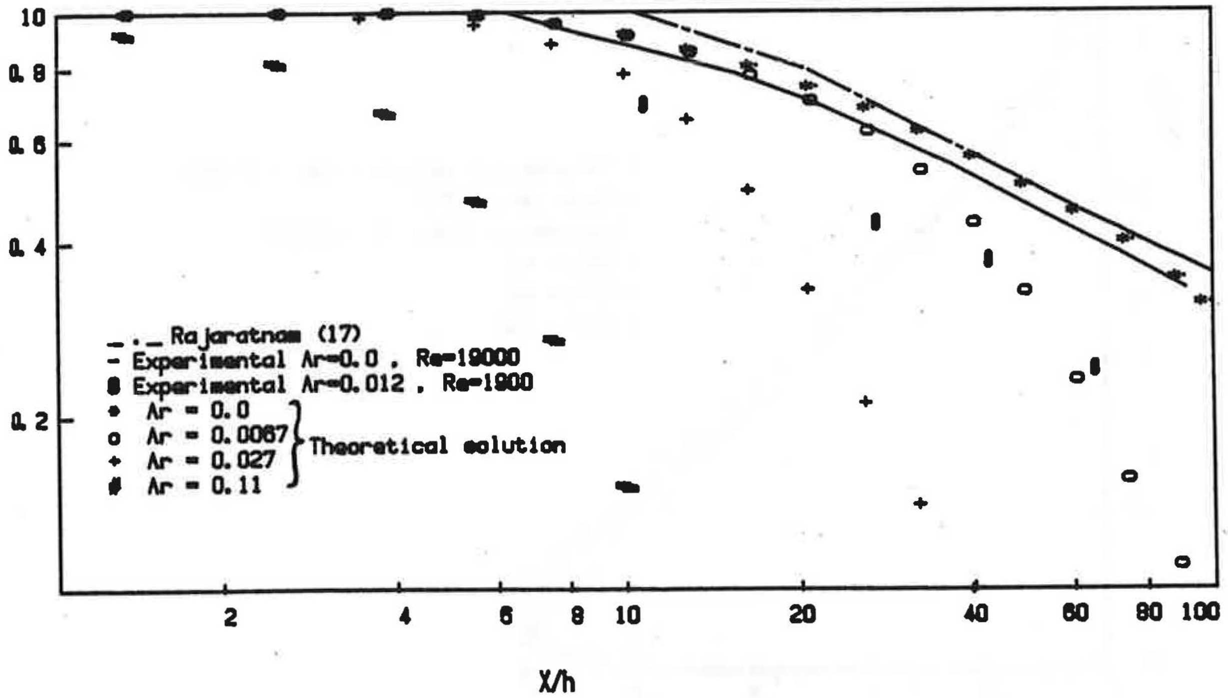


Fig 3 Decay of maximum velocity for different Archimedes numbers

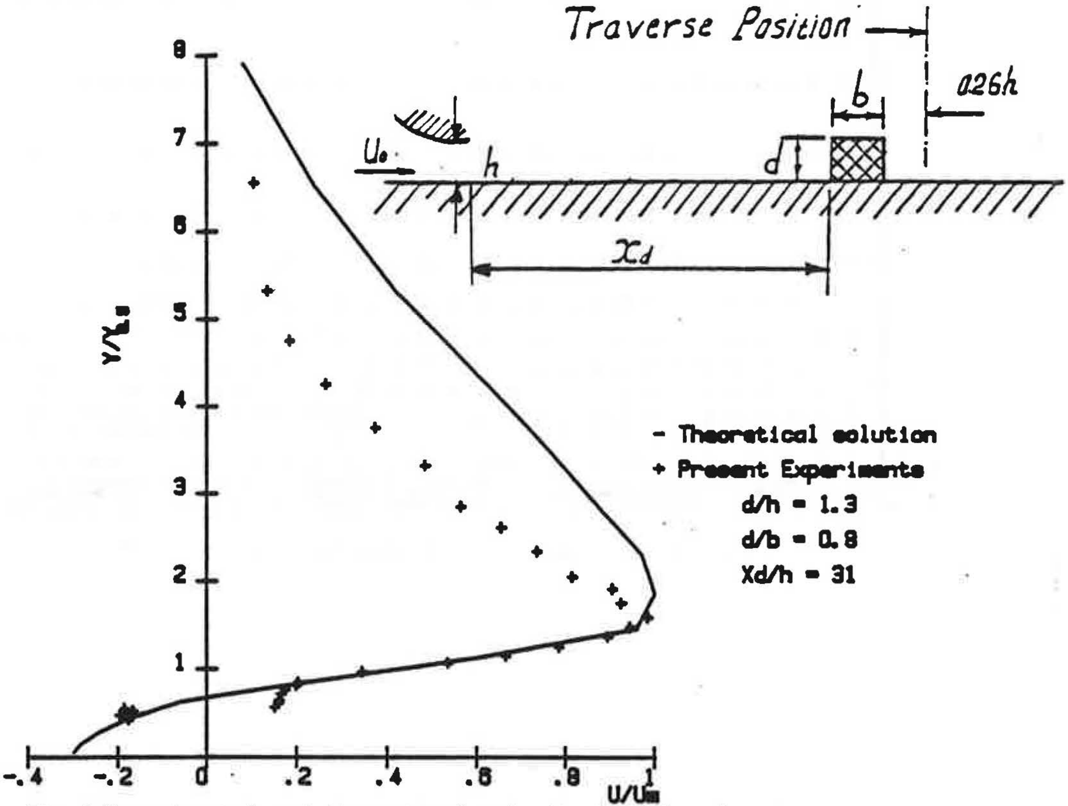


Fig 4 Experimental and theoretical velocity profiles downstream of an obstacle with a reattached jet

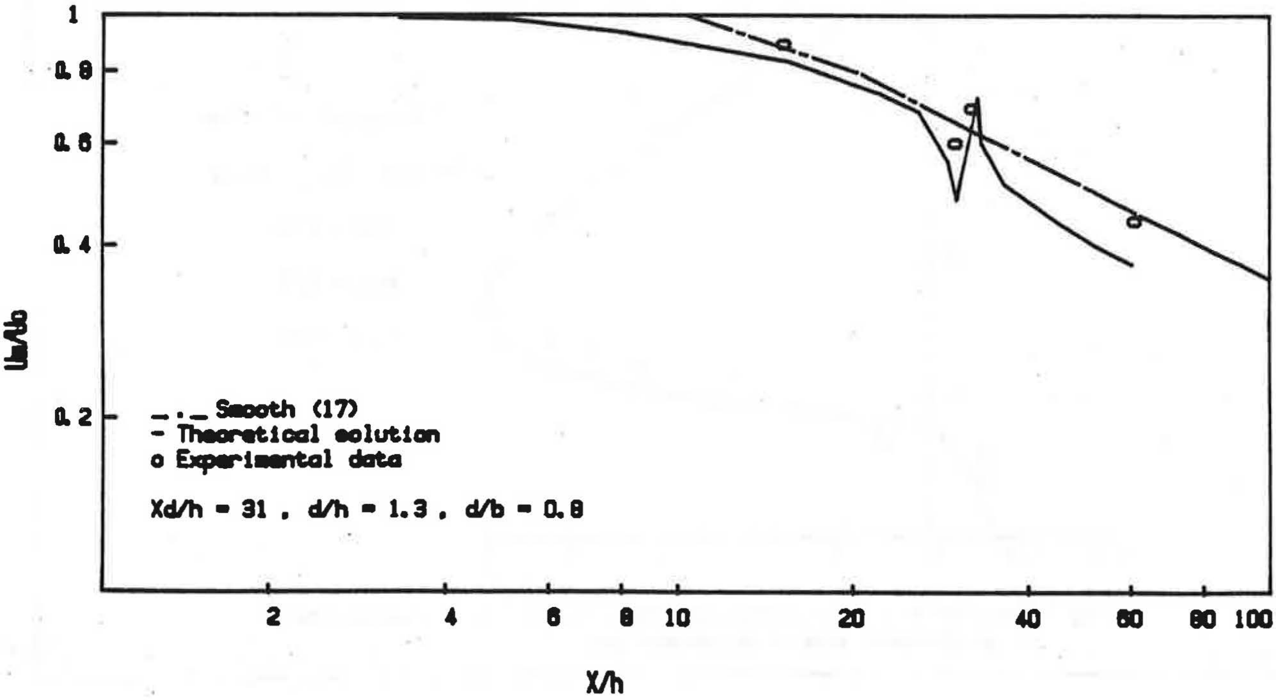


Fig 5 Decay of maximum velocity for a reattached jet

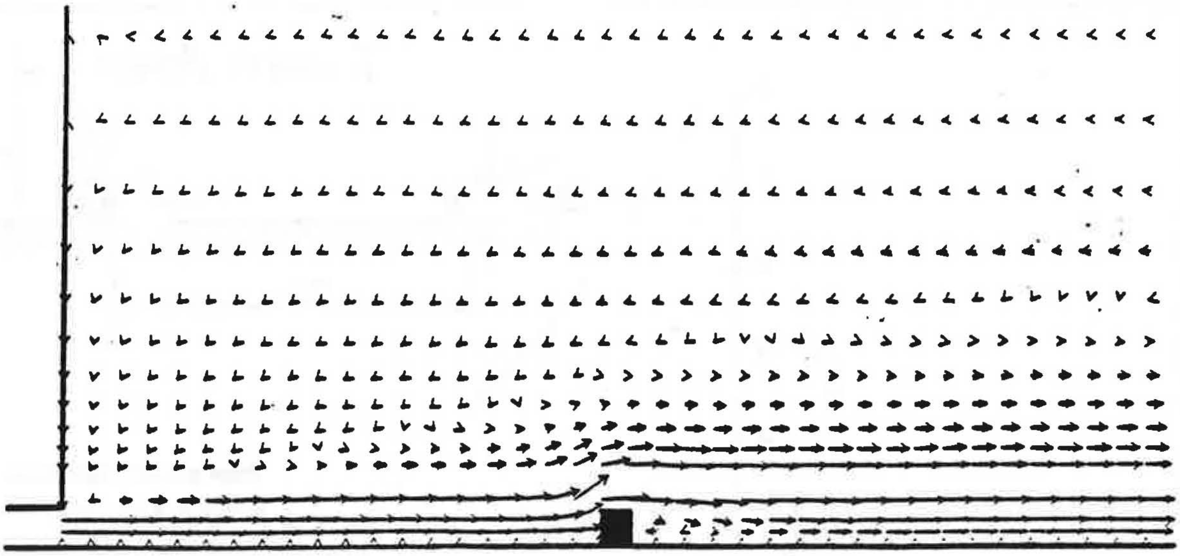


Fig. 6 Velocity vectors over an obstacle for reattached flow

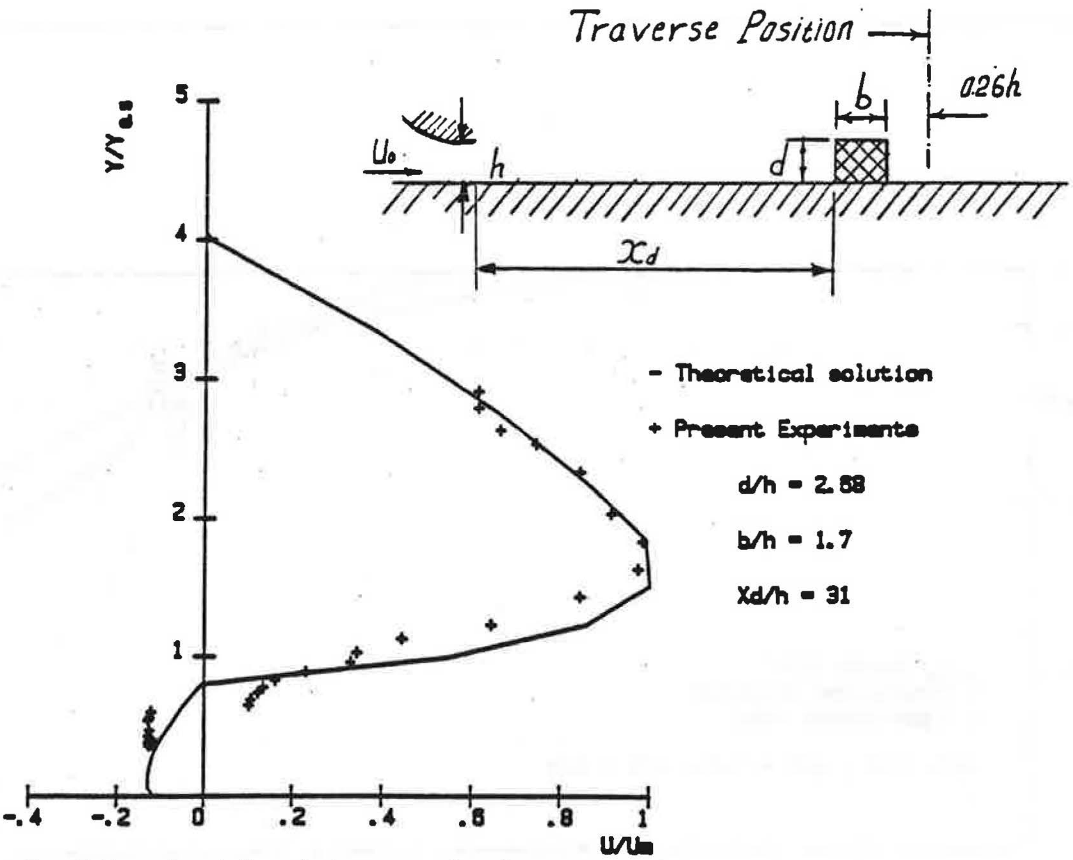


Fig 7 Experimental and theoretical velocity profiles downstream of an obstacle with a separated jet

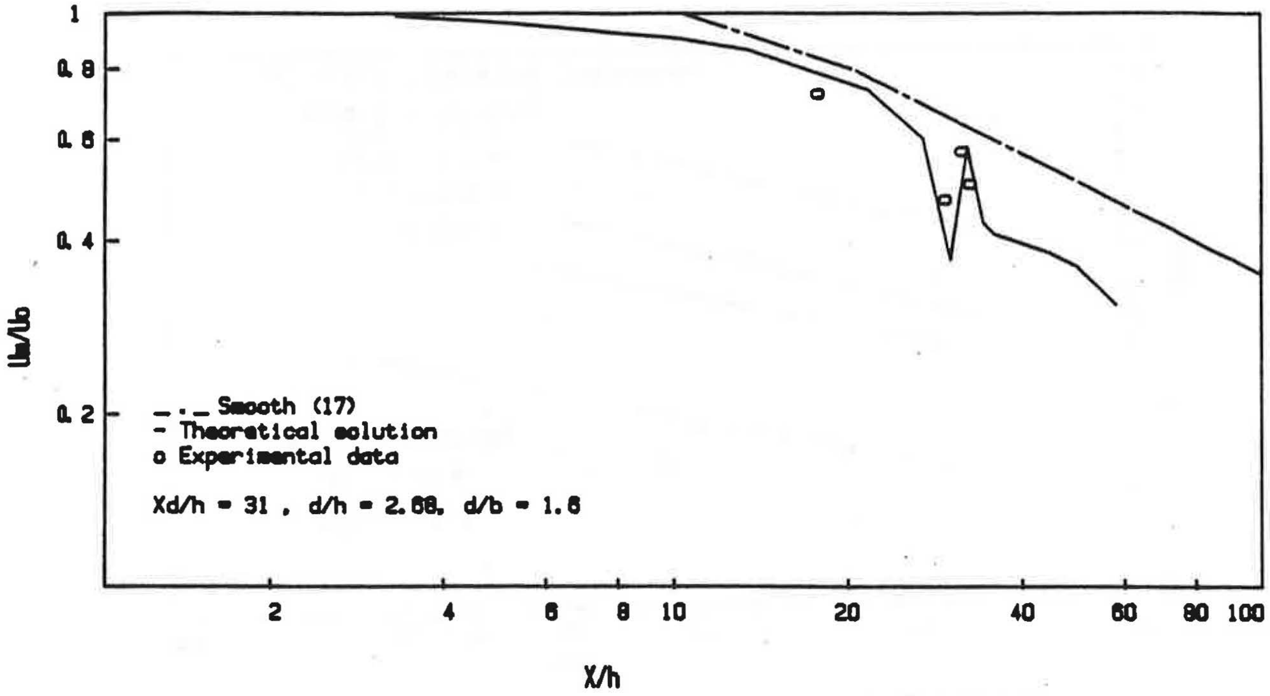


Fig 8 Decay of maximum velocity for a separated jet

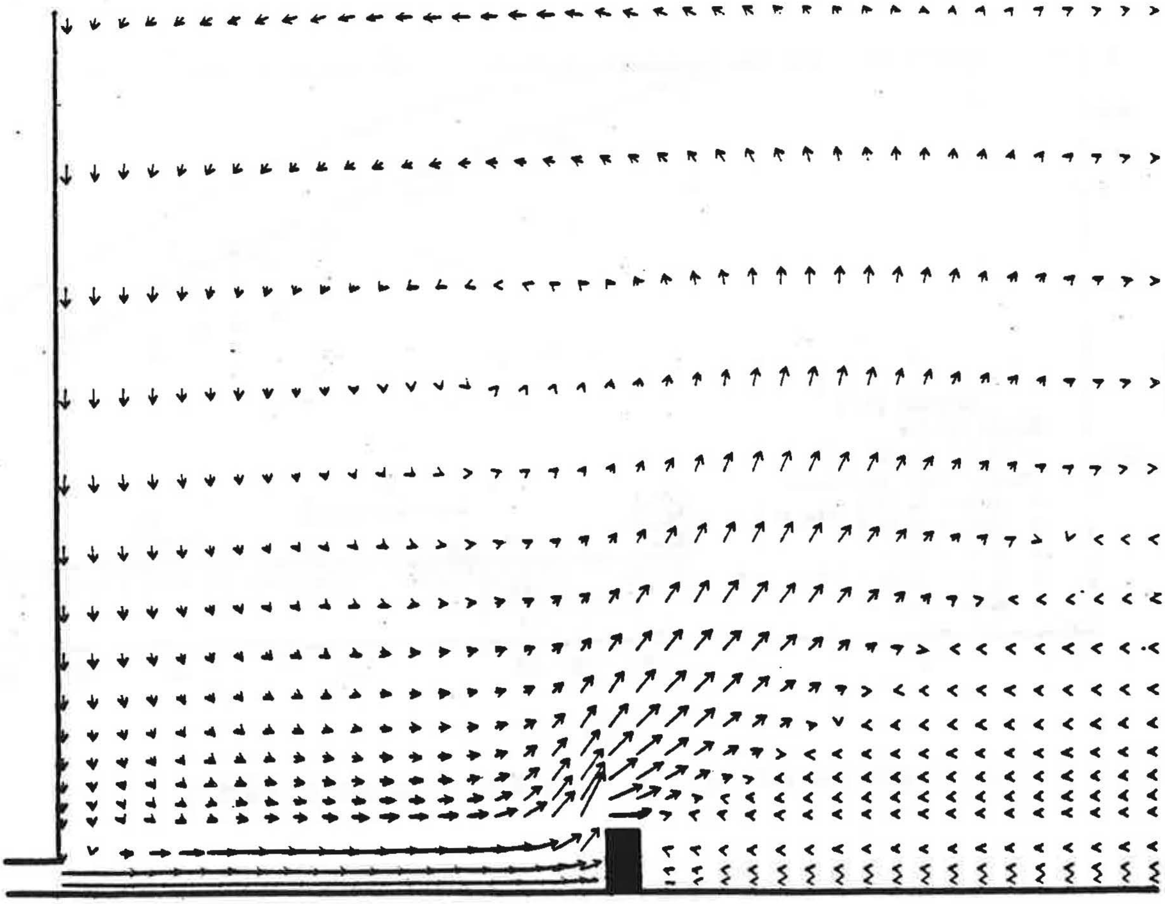


Fig. 9 Velocity vectors over an obstacle for separated flow

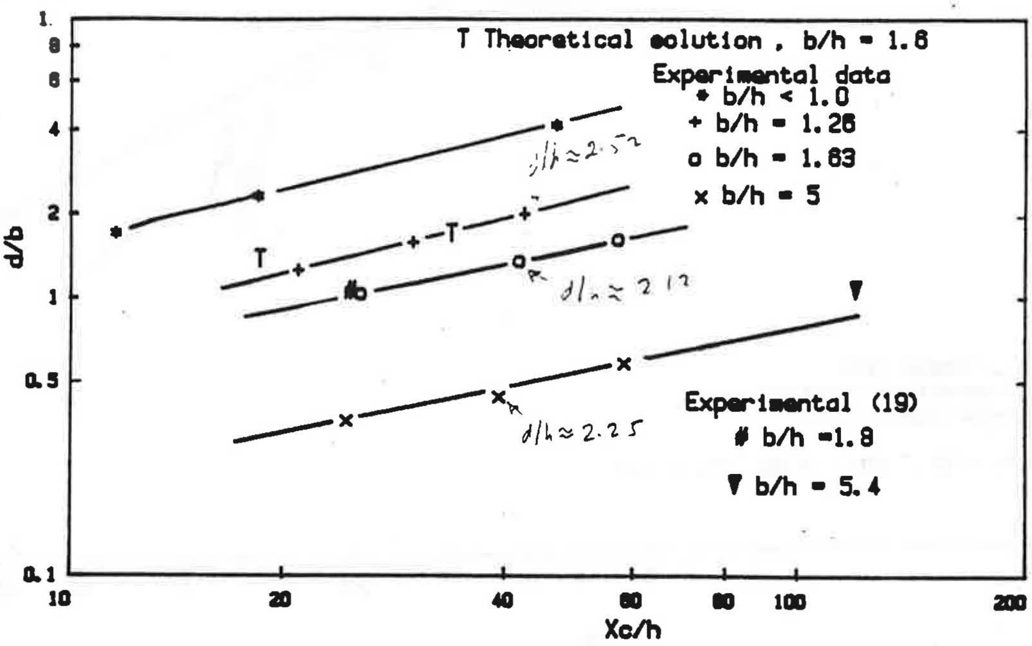
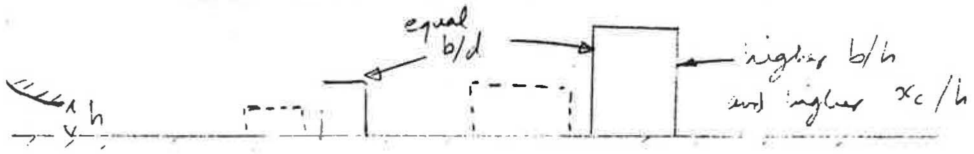


Fig 10 Effect of obstacle dimensions on the critical distance



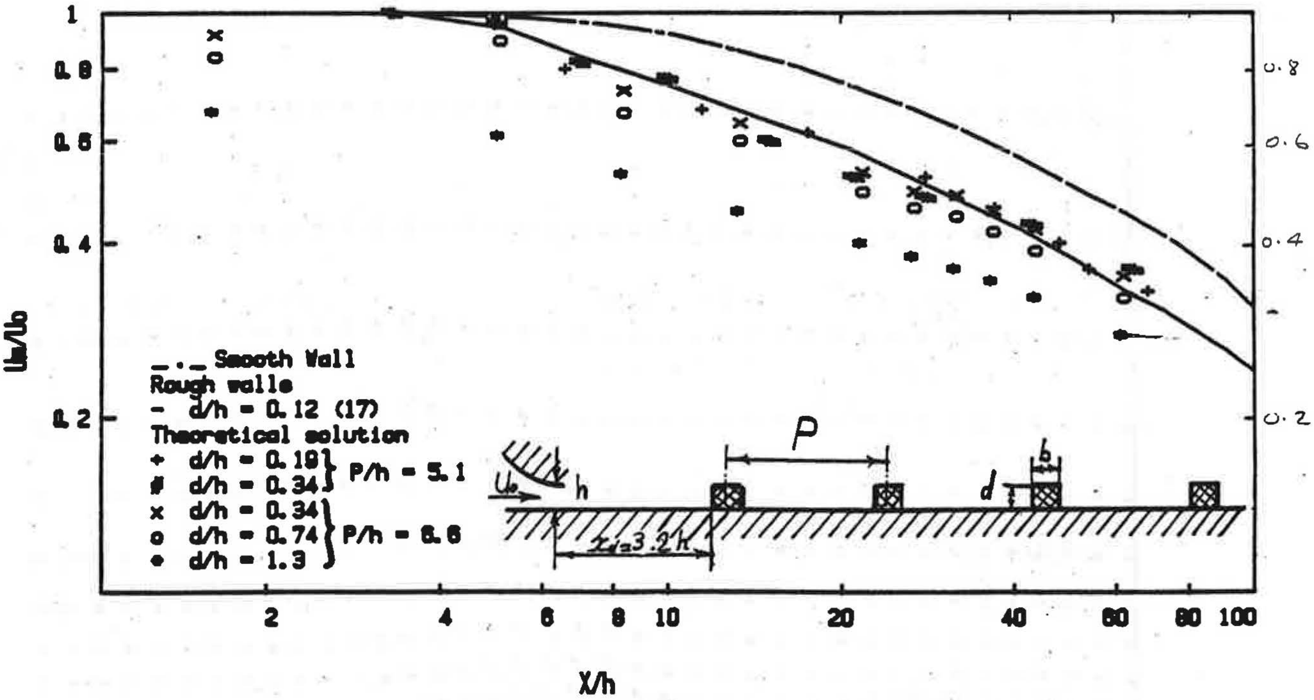


Fig 11 Effect of wall roughness on the decay of maximum jet velocity