The Trajectories of Large Fire Fighting Jets

A. P. Hatton† and M. J. Osborne‡

This article describes a computer simulation of the trajectories of large water jets which allow the effects of changes in initial velocity, elevation, nozzle diameter, and head and tail winds to be examined. The rather limited information on aerodynamic drag of large jets obtained by other workers is used. The predicted trajectories compare well with the limited data available in the literature.

The results also show that for a given flow rate an optimum pressure, and hence an optimum nozzle diameter, exists for maximum throw distance which has important implications for the design of the whole system including the pumps. The optimum elevation in still air lies in the range 30–40°. Wind effects are shown to be very important.

NOTATION

- d Initial jet diameter
- F Froude number $V_0/\sqrt{(gd)}$
- g Gravitational acceleration
- k Drag constant
- n Drag index
- t Time
- V Velocity relative to the air
- V_0 Initial relative velocity
- V_x Component of relative velocity in the x direction
- V_y Component of relative velocity in the y direction
- W Wind speed
- η Vertical jet efficiency
- θ Angle of jet to horizontal

INTRODUCTION

Large fire fighting water jets are required for support vessels for offshore oil rigs. The requirement is to project large quantities of water over the maximum possible distance. The question arises as to whether this is best done by a moderate size jet at high speed or with a larger jet at a lower speed. Very little systematic work has been done on large water jets in contrast to the very extensive literature which exists on fuel sprays and small jets. For a given discharge quantity, it is of considerable interest to know the distance achieved by the jet and how this is affected by such parameters as elevation, pressure, nozzle diameter, and wind velocity.

In this article a computer simulation is described, based on available drag information, which enables the effects of changes in the above parameters to be observed. Although the absolute magnitudes of the throw distances obtained must be treated with some caution, it is reasonable to accept the relative changes caused by changes in the above parameters. A successful simulation could also form the basis of a control system since there is some interest in compensating for wind changes and ship motion to keep the jet on target. Comparison with the very limited available measurements of jet trajectories will be shown to be good.

Received 25 October 1978 and accepted for publication on 12 December 1978.

† Reader in Heat Transfer, Department of Mechanical Engineering, UMIST, Manchester.

‡ Managing Director, Knowsley Engineering Ltd, Trafford Park, Manchester.

SURVEY OF THE EARLIER WORK

The first extensive work on fire streams was published by Freeman (**1**) who was mainly concerned with nozzle discharge coefficients. More recently, a comprehensive series of tests was carried out by Rouse *et al.* (**2**) on behalf of the US Coast Guard.

A great deal of work has been carried out on turbulent jets and a correlation of the break-up distance in terms of the Weber number was proposed by Phinney (**3**) who, in addition to his own experiments, used data by Chen and Davis (**4**) and Grant and Middleman (**5**). A similar correlation was also given by Miesse (**6**). In all these studies, however, the jets were of small diameter (maximum 0·75 in, 19 mm) moving at low speeds and attempts to apply these correlations to the limited data for large jets such as those used by Rouse (maximum 3 in, 76·2 mm diameter) do not succeed. This is not surprising as the surface tension force becomes of decreasing importance as the jet diameter increases. In a large jet the turbulent eddy sizes are much greater and more easily disrupt the jet surface smoothness.

Arato, Crow and Miller (**7**) carried out experiments on a vertical jet with application to fountains. They showed that the efficiency of the jet, defined as the height achieved divided by the head at the nozzle inlet, correlated well as a function of the initial Froude number $V_0/\sqrt{(gd)}$. From their results it is possible to obtain drag coefficient information and this was done to carry out the simulations to be described in this work.

Hoyt and Taylor (**8**) carried out extensive tests on a large variety of nozzle shapes and concluded that nozzle shape does not have an important influence on throw distance. Arato *et al.* and Rouse *et al.* had reached a similar conclusion but all these authors are in agreement that it is of the highest importance to eliminate swirl, to obtain a uniform velocity profile and to reduce turbulence at the nozzle entry in order to achieve maximum jet throw distance.

ANALYTICAL BASIS OF THE SIMULATION

The problem was solved using the equations of motion of a projectile. The equations were expressed in velocities relative to the wind, solved by a step by step method and converted back to absolute values at the end of each step.

The following assumptions were made.

(1) The drag force is given by kV^n per unit mass. This is

0142-727X/79/0300-0037$02.00

the common form of drag law used in turbulent flows with the index n equal to 2 or a value close to 2.

(2) The drag constant k is a function of the initial Froude number F based on the initial relative velocity, that is, k is a constant for a given jet but varies with change of the nozzle diameter and initial velocity. In fact, because large jets steadily lose mass by droplet separation from the surface, this can only be reasonable for the initial part of the flow. The value of k must increase as the jet breaks up. It is hoped to include this effect in future simulations.

(3) The values of k as a function of F are the same as those in the vertical jet tests of Arato *et al.* The method of obtaining the relationship between k and F will be discussed in the next section.

The equations solved were

$$\frac{dV_x}{dt} = -kV^n \cos\theta \tag{1}$$

$$V_x = \frac{dx}{dt} \tag{2}$$

$$\frac{dV_y}{dt} = -g - kV^n \sin\theta \tag{3}$$

$$V_y = \frac{dy}{dt} \tag{4}$$

$$\cos\theta = V_x/V \tag{5}$$

$$\sin\theta = V_y/V \tag{6}$$

$$V = (V_x^2 + V_y^2)^{1/2} \tag{7}$$

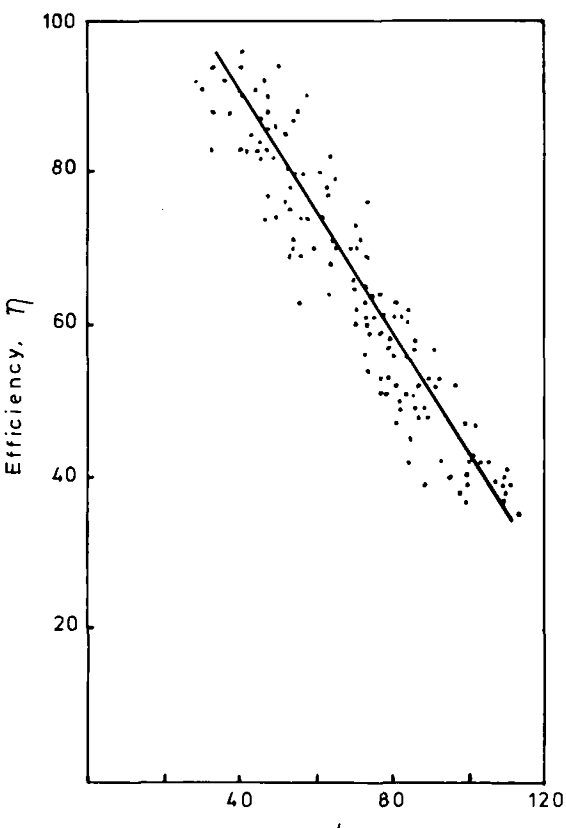

Fig. 1. Vertical jet efficiency versus Froude number F (reproduced from Arato *et al.* (7))

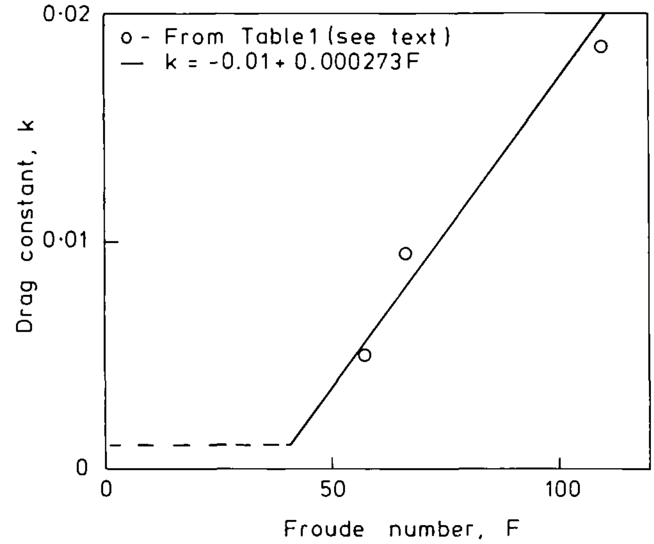

Fig. 2. k–F Relation used in simulation

The velocities in the above equations are all relative to the wind. The initial relative velocity was obtained by vector subtraction of the horizontal wind velocity from the absolute initial jet velocity. This implies the additional assumption that the drag law for the velocity component normal to the jet is the same as that for the component along the jet and clearly this is a feature which could be improved should data for drag forces caused by cross winds become available. The equations were solved by a Runge–Kutta step by step method with an accuracy control on the step lengths. At each step the true velocity was calculated together with the absolute trajectory values of x and y.

Care was required to ensure that the drag constant k did not fall outside the limits of Figs. 1 and 2. Some difficulty was also found with the signs of $\cos\theta$ and $\sin\theta$ since the jet direction, depending on wind strength, can lie in any of the four quadrants (that is, $0 < \theta < 360°$). In very strong winds, for example, the jet, at high elevation, may be swept backwards. On the UMRCC Cyber 72 interactive computer system the central processor time used was 4·5 s for a single trajectory.

DETERMINATION OF k

If it is assumed that $n = 2$ in eqs. (1) and (2), then it can be shown that for a vertical jet in still air the jet efficiency is given by

$$\eta = \frac{g}{kV_0^2} \ln \left| 1 + \frac{kV_0^2}{g} \right| \tag{8}$$

Arato *et al.* give values of η versus F for such a jet and their results are reproduced on Fig. 1. They used three nozzle diameters of 1·45 in (36·8 mm), 1·0 in (25·4 mm), and 0·667 in (17·45 mm). They also quote the head on the nozzle from which V_0 can be obtained. It is therefore possible to solve eq. (8) for k and to obtain corresponding values of F. Three cases are given in Table 1 and shown on Fig. 2. Because the values are subject to some observational error it was considered sufficiently accurate to represent them by a straight line. The line used was

$$k = -0{\cdot}01 + 0{\cdot}000273F \tag{9}$$

Table 1

Results of Arato et al. (7) and calculated values of k

Experiments of Arato *et al.*				Value of k calculated from eq. (8)	Value of η calculated from program ($\theta = 85°$)
η	d (m)	V_0 m/s	F		
0·41	0·01745	45	108	0·0185	0·394
0·67	0·0254	32·7	65·5	0·0095	0·71
0·78	0·0368	34·2	56·9	0·0050	0·72

It will be seen that k becomes zero at $F = 37$ and would be negative for values of F below 37. This, of course, is not possible and the program was arranged to put k at the small value of 0·001 for $F < 37$ and to abandon the calculation should the value of F be greater than 120 since the experiments did not extend beyond this value.

RESULTS

(1) Comparison with Available Experimental Results

It was first necessary to ensure that the simulation agreed with the results of Arato *et al.* quoted in Table 1. The solution was therefore carried out for these cases with $\theta = 90°$ and $\theta = 85°$. The height achieved was virtually the same for both angles and the jet efficiencies so obtained are given in the final column of Table 1 and are seen to be in good agreement with the observed values. This of course merely confirms that the empirical form used for k reproduces the results from which it was obtained. The differences are due to scatter of the experimental results.

Rouse's results give the trajectories of jets from various nozzle types and at different pressures and elevations. However, he shows the trajectories only up to the position where the jet remains coherent. Figure 3 shows some comparisons with four trajectories given by Rouse

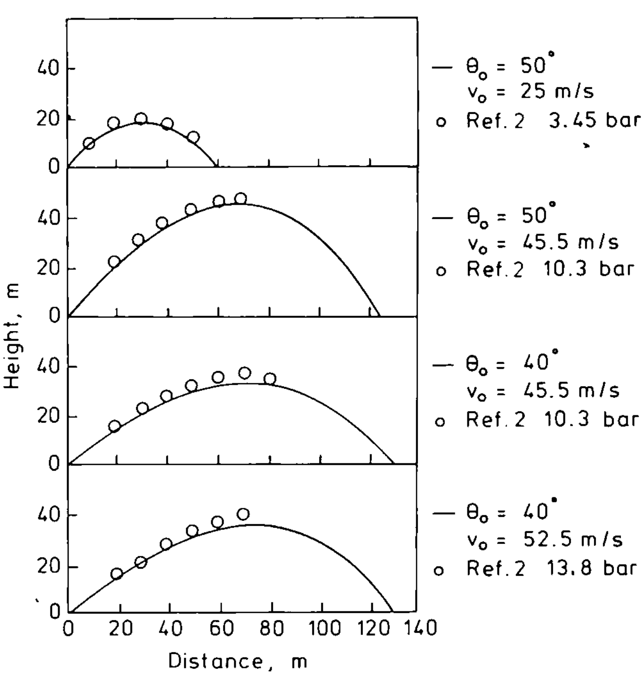

Fig. 3. Comparison of prediction with experiments of Rouse *et al.* (2). Nozzle dia. 3 in (76·2 mm)

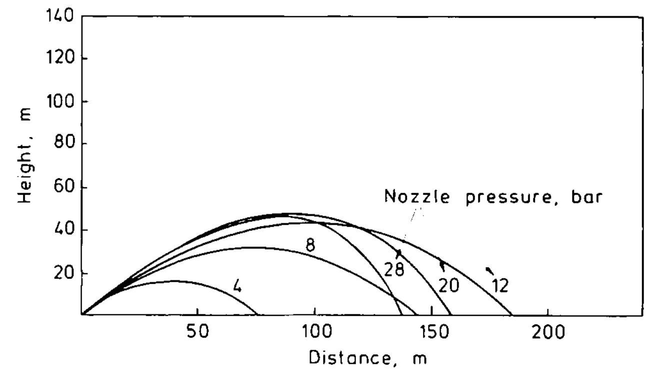

Fig. 4. Jet trajectories in still air $\theta_0 = 40°$, flowrate 2400 m^3/h

who measured the trajectories photographically and the agreement is seen to be good. However, Rouse quotes the monitor base pressure in his results whereas the simulation requires the nozzle pressure which makes the direct comparison slightly uncertain. One significant result of Rouse *et al.* is that, for a jet of 3 in (0·076 m) diameter at a fixed elevation, increases of pressure above 200 lb/in^2 (14 bar) do not give increase of throw distance. Indeed only very small gain results from pressures above 150 lb/in^2 (10·3 bar). This result was exactly predicted by the simulation and will be discussed in more detail below.

(2) Results of Prediction in Still Air

A series of simulations was carried out with a fixed delivery quantity. A few preliminary runs showed that the maximum throw distance was obtained with elevations in the regions of 30 degrees to 40 degrees. For a fixed flow quantity the nozzle diameter is reduced as the pressure increases.

Figure 4 shows jet trajectories for a flow rate of 2400 m^3/h and a fixed elevation of 40 degrees. The nozzle pressure was varied from 4 bar to 28 bar and the corresponding nozzle diameter changed from 0·1732 m to 0·1064 m. Since the drag constant increases with both increase in velocity and reduction of diameter, the effects of drag become more pronounced at high Froude number and produce a reduction of maximum throw.

This is shown clearly on Figs. 5 and 6, and it is seen that for each flow rate an optimum pressure exists to achieve maximum throw distance. The magnitude of the throw distances themselves should be regarded with some caution due to break-up in the final stages. However, such experimental work as does exist, particularly that of Rouse, appears to support this result. The result is important in connection with large fire pumps since large pressures demand high power and clearly there is no advantage to be gained above the optimum.

Figure 7 shows the same effect for a nozzle of fixed diameter (0·125 m). As the pressure increases in this case, of course, the flow rate rises. However, there is again no advantage to be gained in still air by using pressures of above approximately 13 bar. Above this pressure the drag force kV^2 increases, due to increase of both k and V, at such a rate as to reduce the total throw distance.

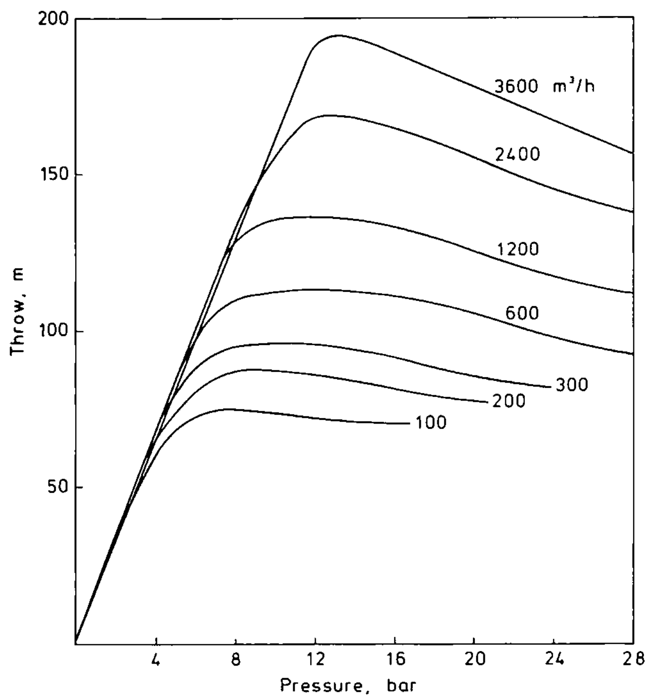

Fig. 5. Maximum throw distances for given flowrates in still air $\theta_0 = 30°$

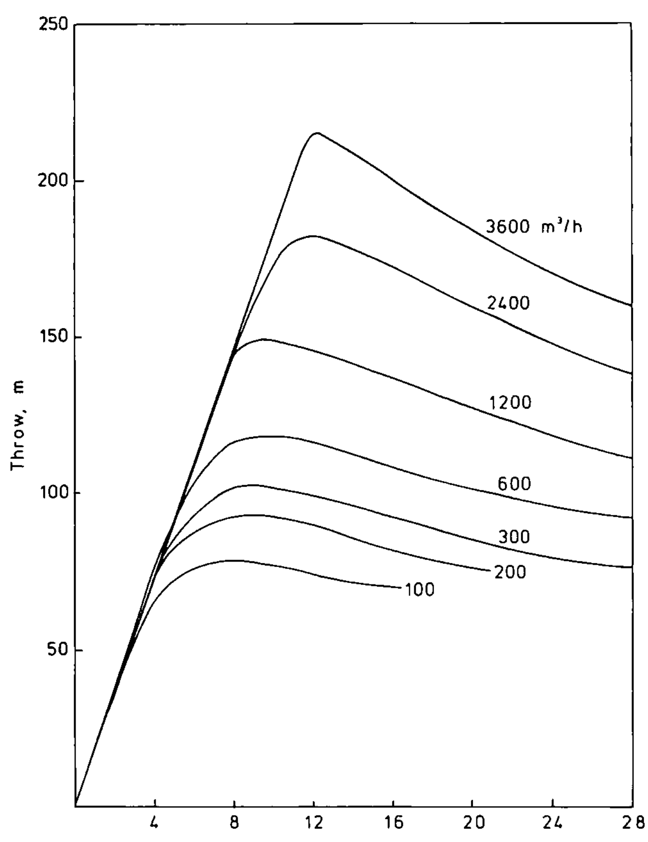

Fig. 6. Maximum throw distances for given flowrates in still air $\theta_0 = 40°$

(3) Effects of Head and Tail Winds

It is well known that jets for fire fighting are strongly influenced by wind. Obviously a fire fighting vessel should be so positioned as to make the best possible use of the wind if maximum throw distance is required. Figure 8 shows the effect on the trajectory of a jet of head and tail winds of 10 m/s (approximately force 5 on the Beaufort Wind Scale). It is seen that the simulation bears out experience by showing an increase of throw distance of $\times$ 2·4 with the wind astern compared with the wind ahead.

Figure 9 also shows that there is some advantage in having higher pressure available and rather surprisingly it is with the wind astern that the optimum pressure becomes higher (16 bars for 3600 m^3/h). Figure 10 shows the optimum pressures for zero wind velocity and a 10 m/s tail wind.

CONCLUSIONS

The use of a computer simulation of fire fighting jets has been shown to be useful in assessing the response of the jet to differing operating parameters. In particular it has been shown that an optimum pressure, and hence an optimum nozzle diameter, exists for a given flow quantity. From this information it is possible to optimize the

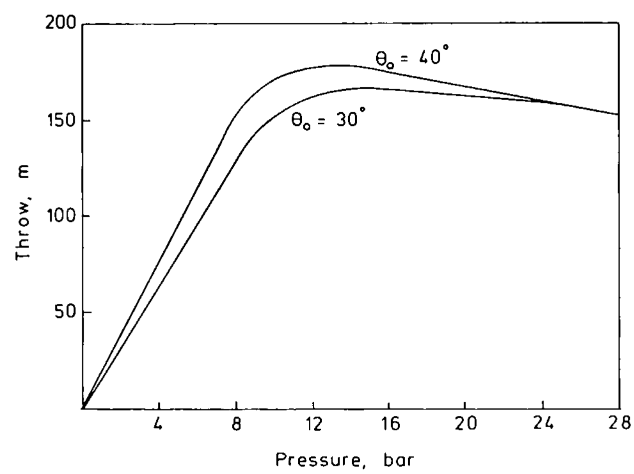

Fig. 7. Maximum throw distance of a fixed diameter nozzle (0·125 m) in still air

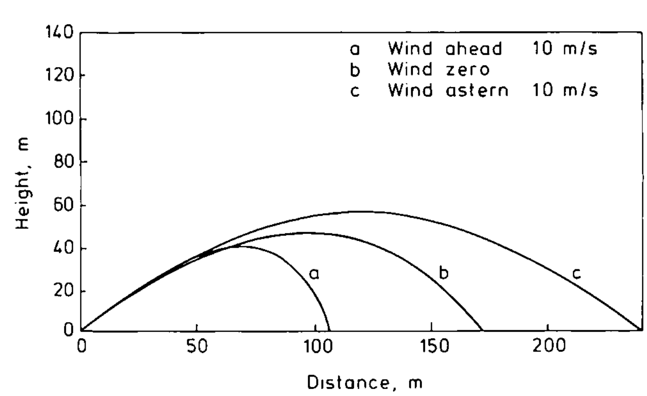

Fig. 8. Effect of force 5 wind on jet trajectory $\theta_0 = 40°$, flowrate 2400 m^3/h, pressure 16 bar

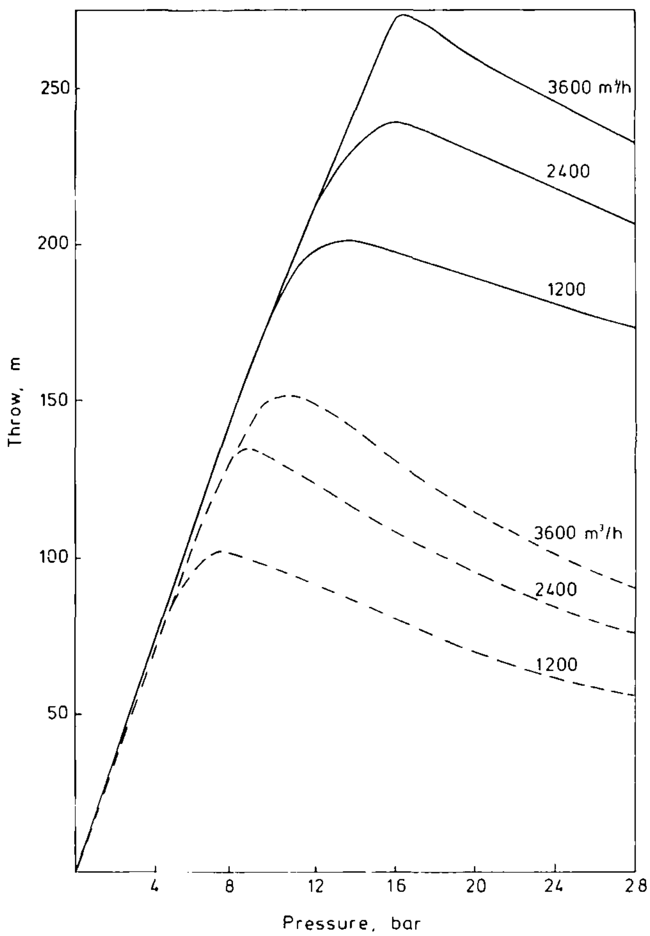

Fig. 9. Effect of wind (10 m/s) on maximum throw distance for given flowrates $\theta_0 = 40°$: —— Wind astern; - - - Wind ahead

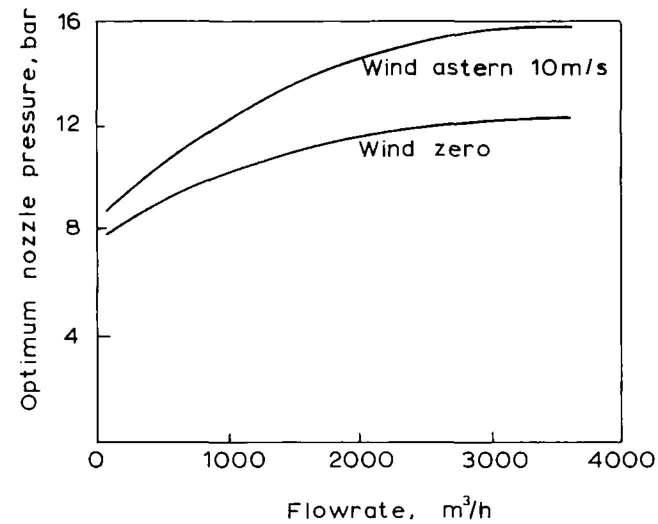

Fig. 10. Optimum nozzle pressures for maximum throw distance

designs of both the monitor and pump required for a given duty.

The predicted trajectories correspond quite accurately with experimentally observed results, certainly to the apogee, the highest point reached in the trajectory, and probably for that part of the jet where it remains coherent, namely, up to ranges of 10–20 per cent beyond the apogee. Thereafter the jet may decay more rapidly or be more subject to wind effects than the simulation provides.

The simulation will hopefully be developed in the future to include a better description of the final stages of the trajectory and also to include cross wind effects.

ACKNOWLEDGEMENTS

The authors are grateful to Dr C. M. Leech and Dr M. R. F. Heikal for assistance in the numerical solution of the equations and the programming techniques used.

REFERENCES

(1) FREEMAN, J. R. 'Experiments Relating to Hydraulics of Fire Streams', *ASCE Trans.* 1889, **21**, 303–461
(2) ROUSE, H., HOWE, J. W., and METZLER, D. E. 'Experimental Investigation of Fire Monitors and Nozzles', *Proc. ASCE.* 1951, **77**, 29
(3) PHINNEY, R. E. 'Breakup of a Turbulent Liquid Jet in a Low-Pressure Atmosphere', *A.I.Ch.E.J.* 1975, **21**, 996–999
(4) CHEN, T. F., and DAVIS, J. R. 'Disintegration of a Turbulent Water Jet', *Proc. ASCE.* 1964, **HY1**, 175
(5) GRANT, R. P., and MIDDLEMAN, S. 'Newtonian Jet Stability', *A.I.Ch.E.J.* 1966, **12**, 669
(6) MIESSE, C. C. 'Correlation of Experimental Data on the Disintegration of Liquid Jets', *Ind. Engng Chem.* 1955, **47**, 1690
(7) ARATO, E. G., CROW, D. A., and MILLER, D. S. 'Investigations of a High Performance Water Nozzle', *BHRA Res. Report.* June 1970, 1058
(8) HOYT, J. W., and TAYLOR, J. J. 'Effect of Nozzle Shape and Polymer Additives on Water Jet Appearance', *ASME Paper.* No. 77-FE-16, 1977