LABORATORY MODELLING OF NATURAL VENTILATION FLOWS DRIVEN BY THE COMBINED FORCES OF BUOYANCY AND WIND

Gary R. Hunt & Paul F. Linden.
Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Silver Street, Cambridge, CB3 9EW.

This paper describes an innovative experimental technique that accurately reproduces natural ventilation flows, driven by the combined effects of stack and wind, at small scale in laboratory models of rooms or buildings. This technique provides a powerful tool for examining the performance of naturally ventilated buildings at the design stage as it may be used to predict *quantitatively* ventilation flow rates and temperature stratification under a wide range of climatic conditions.

Following the success of saline modelling techniques to model stack-driven flows these techniques have now been extended to consider building ventilation under the combined effects of stack and wind. A full description of the experimental technique is given and it is shown that dynamical similarity is achieved between small-scale laboratory flows driven by combined stack and wind effects and those at full scale. Scaling laws used to transform the quantities from small scale to those at full scale are given.

INTRODUCTION

Over the past few years our understanding of natural ventilation has been significantly advanced and improved by new research using laboratory models [1]. The aim of this research has been to gain an improved understanding of the physics of air flows in buildings and from this new understanding of these complex processes, develop simple design guidelines for producing energy-efficient and reliable natural ventilation systems for buildings.

In order to examine the performance of naturally ventilated buildings at the design stage laboratory studies have been carried out on small-scale models in water tanks. A small-scale (typically 1:20 - 1:100) model of a room or building is constructed from transparent Perspex and immersed in a large tank of fresh water. The large body of fresh water which surrounds the model represents the outside air. The scale model need not be an exact replica of the test building although the essential features controlling the ventilation must be accurately represented. This simplification requires an understanding of the physical principles and usually involves ensuring that the openings are accurately modelled [2]. The stack effect is simulated by adding salt solutions of different densities to the model to represent temperature differences between the internal and external air. Salt solutions are more dense than fresh water, and hence, buoyancy forces are generated which are analogous to those produced by warm and cool air masses. To represent a space which is initially at a different temperature than its surrounding the model is filled with a salt solution; the denser the salt solution the greater the temperature difference simulated. Salt solutions may also be added continuously to represent inputs of heat associated with internal and solar gains. As brine is denser than fresh water the buoyancy forces act downwards, in contrast to those generated by inputs of heat, and therefore, for the case of a warm space the model is inverted and viewed through an inverted video camera so that the dense salt solution appears to 'rise' in the model in the same way as warm air rises when surrounded by cooler air. Clearly, by filling the large environmental tank with brine and using fresh water in the model it is possible to create buoyancy forces which act upwards. It is purely for economical reasons that fresh water is used to represent the ambient air.

When modelling natural ventilation flows in the laboratory, water, rather than air, is used as the working fluid for three reasons. First, the flow is easily visualised using shadowgraph imagery (which shows density variations within the flow) and dyes to colour different regions of the flow. Second, quantitative measurements of flow velocity and density fields inside the model can be made using sophisticated digital image processing techniques. These techniques give synoptic measurements of the flow within the model and show, for example, how the stratification evolves as a result of changes in the ventilation rate. Third, accurate *quantitative* predictions can be made about the flow at full scale as the effects of friction and diffusion (which are very small for typical ventilation flows and are characterised by large Reynolds and Péclet numbers, respectively) are accurately modelled at small scale, see [2]. The use of salt allows high Reynolds numbers to be achieved in small models; water is a less viscous fluid than air and it is possible to obtain large density differences using high salt concentrations which give large flow velocities. This dynamic similarity means that quantitative information can be obtained from small-scale models and extrapolated to full-sized buildings.

APPLICATION TO BUILDING VENTILATION

The saline modelling technique has been used to predict the performance of a number of naturally ventilated buildings at the design stage. Case studies have so far been restricted to stack-driven flows and have included the Cable and Wireless building, Coventry [1], the Engineering School at De Montfort University [3] and the atrium section of the Chelsea and Westminster Hospital in London [4]. In each of these studies the technique was used to assess the suitability of the proposed design to provide adequate natural ventilation and to identify any problem areas within the space. By measuring density and velocity within the model the corresponding air temperatures and ventilation flow rates within the test building were predicted, and hence, the ventilation performance assessed. Quantitative comparisons between measurements taken in the model and those at full scale have confirmed that large-scale flows are accurately represented at small scale, see for example [3].

The laboratory technique provides a rapid, cost-effective and reliable means of examining air flows within ventilated spaces. In many practical situations, for example, where the building design is complex or consists of multiple interconnecting spaces, it has distinct advantages over alternative methods. In these cases, computational fluid dynamic (CFD) models may be very expensive, both in terms of real time and computational time, especially when design changes are being considered and a range of building geometries needs to be modelled. Questions remain about the validity of these calculations especially when density effects are significant. In other applications, where knowledge of the vertical temperature profile is required, zonal models (although providing mean flow quantities) are not suitable. For example, in naturally ventilated atrium buildings a strong temperature stratification often develops. If the excess heat is not allowed to escape, the upper layer of 'hot' air may descend to the occupied levels of the space resulting in overheating, although the *average* air temperature in the space (as predicted by zonal models) may be much lower and meet the requirements for comfort.

EXTENSION OF THE TECHNIQUE TO INCLUDE WIND EFFECTS

The saline modelling of natural ventilation has so far been restricted to flows driven by the stack effect and unaffected by external pressures. Although powerful in describing 'worst' case scenarios, these conditions do not describe the flows over the majority of the year when the ventilation is driven by combined buoyancy and wind forces. In such cases, the rule-of-thumb advice is either to consider the greater of the two forces or to add the effects of the pressure differences created by the two forces. Such models do not describe the internal conditions of the space in any detail and assume perfect mixing.

In order to overcome these problems and aid our understanding of natural ventilation processes the saline modelling technique has now been extended to include both wind and buoyancy effects. The extension of the technique involves suspending the model in the test section of a large flume tank containing fresh water. The flume generates a horizontal flow, of controllable speed, in its test section, and this simulates a flow of wind around the model. A schematic diagram illustrating the typical experimental set-up is shown in figure 1.

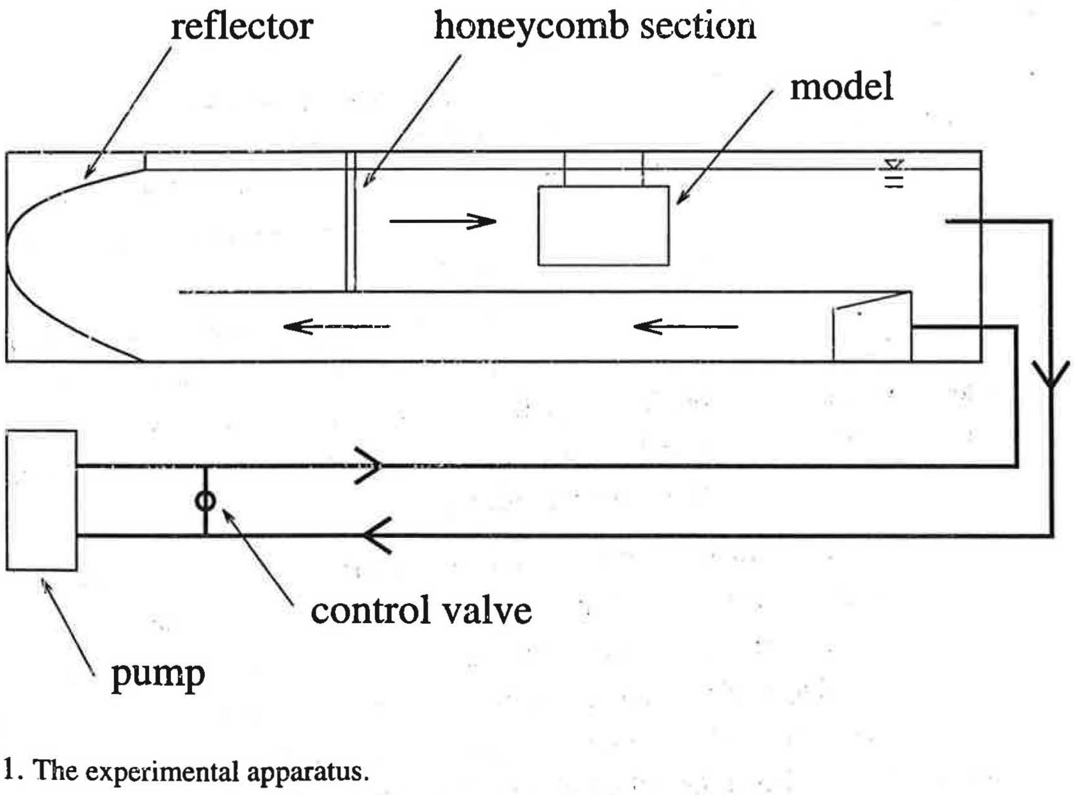

Figure 1. The experimental apparatus.

The flow of water around the model results in a 'wind' pressure drop Δ between the windward and leeward openings which is measured in the experiments using a manometer tube. A simple manometer tube is composed of two transparent sections of L-shaped rigid tubing connected together by a length of flexible tubing. The two ends of the tube are immersed in the flume and positioned level with and adjacent to the upwind and downwind vents of the model, see figure 2. By partially filling the tube with an oil and measuring the displacement X of the oil/water interface in the tube the 'wind' pressure drop

Δ between the openings in the model can be determined from the relationship

$$\Delta = gX(\rho_{oil} - \rho_{water}), \tag{1}$$

where g is the acceleration due to gravity and ρ_{water} and ρ_{oil} denote the density of the water and the oil, respectively.

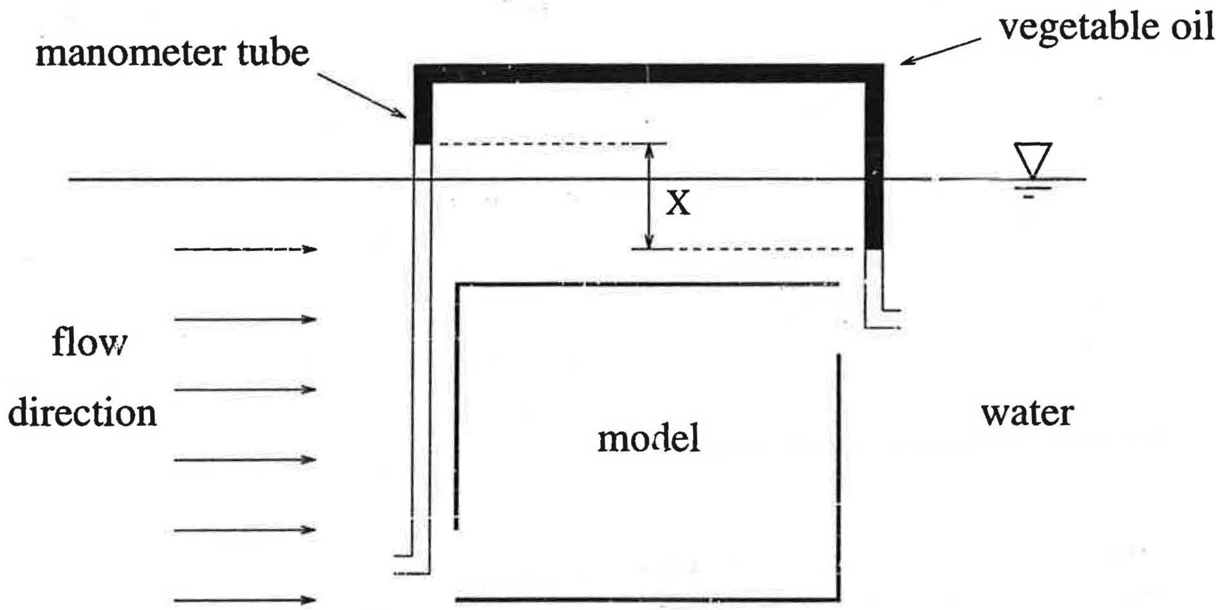

Figure 2. Measuring 'wind' pressure drop using a manometer tube.

The effects of a change in wind direction on the ventilating flow can be examined by varying the angle the model makes with the oncoming stream or by varying the location of the openings. In each case the 'wind' pressure drop can be measured using the manometer tube, and hence, the 'wind' pressure envelope around the model determined. By varying the density difference between the fluid contained in the model and its surroundings and the 'wind' pressure drop, the fluid flow within the space can be examined for the wide range of conditions experienced over the year [5,6,7].

EXTRAPOLATING FROM THE LABORATORY TO FULL SCALE

Flows driven by combined buoyancy and wind forces are dependent upon the wind speed and direction, the geometry of the building, the density difference $\Delta\rho$ between the internal and external environment and upon the area and location of the ventilation openings. The relative magnitudes of the wind-produced and buoyancy-produced velocities is measured as a Froude number, and hence, as the wind pressure drop Δ is related to the square of the wind speed U_f, the Froude number Fr may be defined as

$$Fr = \sqrt{\frac{\Delta}{\rho g' H}}, \tag{2}$$

where H is the vertical distance between the windward and leeward openings, $g' = g\Delta\rho / \rho$ is the reduced gravity and ρ is the density of the ambient fluid. For $Fr < 1$ buoyancy effects dominate the air flow within the building and the effect of the wind on the ventilation flow is small, whereas for $Fr > 1$ the driving produced by the wind is dominant although buoyancy forces may still have a significant effect on the vertical temperature profile within the space, see [6]. It has been shown by [2] that the effects of friction and diffusion are accurately modelled at small scale when fresh and salt water solutions are used to create density differences. Dynamical similarity between wind and buoyancy forces in the model and those at full scale is achieved when the Froude numbers at small and full scale are matched. This requires

$$\left(\frac{\Delta}{\rho}\right)_f = \left(\frac{\Delta}{\rho}\right)_m \frac{g'_f H_f}{g'_m H_m}, \qquad (3)$$

where the subscripts f and m denote quantities at full scale and those used in the model. Dynamical similarity is achieved at small scale in the laboratory as high salt concentrations can be used to produce large density differences, i.e. $g'_m >> g'_f$, and as typical wind speeds are far greater than the mean flow speed generated in the flume, hence $(\Delta/\rho)_m << (\Delta/\rho)_f$. Wind pressure drop and wind speed are related by the expression

$$\left(\frac{\Delta}{\rho}\right)_f = \frac{U_f^{\ 2}}{2}\left(C_{pi} - C_{po}\right)_f, \qquad (4)$$

where C_{pi} and C_{po} denote the pressure coefficients at the inlet and outlet, respectively. Hence, by combining (3) and (4) we can relate the 'wind' pressure drop between the openings in the model to the wind speed at full-scale:

$$U_f = \sqrt{\frac{2}{\left(C_{pi} - C_{po}\right)_f}\left(\frac{\Delta}{\rho}\right)_m \frac{g'_f H_f}{g'_m H_m}}. \qquad (5)$$

Thus, the mean flow speed in the flume which simulates a flow of wind past the small-scale model can be converted to an equivalent wind speed past a full-scale building.

Ventilation flows in the laboratory models occur on different time scales to flows in buildings due to the differences in scale, density contrast and wind speed. For flows driven by buoyancy forces alone the relationship between times in the model and real times is given by

$$\frac{t_f}{t_m} = \sqrt{\frac{g'_m H_f}{g'_f H_m}}, \qquad (6)$$

see [2]. When the ventilation flow is driven by the combined forces of buoyancy and wind, see for example [6], the velocities scale as

$$\frac{\text{velocity}_f}{\text{velocity}_m} = \sqrt{\frac{g'_f H_f \pm (\Delta/\rho)_f}{g'_m H_m \pm (\Delta/\rho)_m}} = \frac{\sqrt{g'_f H_f}\sqrt{1 \pm Fr_f^2}}{\sqrt{g'_m H_m}\sqrt{1 \pm Fr_m^2}}, \qquad (7)$$

where the positive/negative sign convention applies when the natural forces of wind and buoyancy act together/oppose one another, respectively. The time scale is set by the ratio of a representative length and velocity scale, and hence, the following relationship between the times in the model and at full scale may be deduced:

$$\frac{t_f}{t_m}=\frac{\sqrt{g'_m H_f}}{\sqrt{g'_f H_m}}\frac{\sqrt{1\pm Fr_m^2}}{\sqrt{1\pm Fr_f^2}}. \qquad (8)$$

By expanding (8) in terms of Fr we obtain the following time scales in the limit of small and large Fr,

$$\frac{t_f}{t_m}=\begin{cases}\sqrt{\dfrac{g'_m H_f}{g'_f H_m}} & \text{for } Fr \ll 1\\[2ex] \dfrac{H_f\sqrt{(\Delta/\rho)_m}}{H_m\sqrt{(\Delta/\rho)_f}} & \text{for } Fr \gg 1.\end{cases} \qquad (9)$$

The ratio of the lengthscales H_f/H_m is set by the model scaling, and hence, when wind forces are dominant compared with buoyancy forces the real and model times are set by the ratio of the wind-induced velocities in the model and at full scale. If the Froude number in the laboratory flow is matched exactly with that at full scale then from (8) we have

$$\frac{t_f}{t_m}=\frac{\sqrt{g'_m H_f}}{\sqrt{g'_f H_m}}. \qquad (10)$$

CONCLUSIONS

An experimental technique has been described which provides a powerful tool for examining the performance of naturally ventilated buildings at the design stage. The technique allows ventilation flows driven by the *combined* forces of wind and buoyancy to be reproduced in small-scale models of buildings in water tanks. Buoyancy forces are generated in the model using salt solutions and fresh water, and wind flow past the building is simulated by immersing the model in a flume tank with a mean flow. 'Wind' pressure drop between openings in the model can be measured using a manometer tube, and hence, wind-induced velocities determined.

Dynamical similarity is achieved between combined buoyancy and wind driven flows at small scale in the laboratory models and those at full scale in buildings. This means that quantitative measurements of density and velocity made at small scale can be extrapolated to full scale, and hence, accurate predictions made about the temperature distribution and air flow rates in buildings. By varying the density of the salt solutions, the mean flow in the flume and the orientation of the model and its openings, the temperature distribution and ventilation flow rates within a building can be predicted for a wide range of climatic conditions. By modelling natural ventilation using this technique air flows in buildings can be clearly visualised at small scale using colour dyes and shadowgraph imagery and this has aided in our developing an improved understanding of these often complex flows and provided a means of transferring this knowledge to a wider audience.

REFERENCES

[1] LINDEN, P.F., LANE-SERFF, G.F. & SMEED, D.A. 1990 Emptying filling boxes: the fluid mechanics of natural ventilation. *J. Fluid Mech.*, **212**, 300-335.

[2] BAKER, N. & LINDEN, P.F. 1991 Physical models of air flows - a new design tool. *Atrium Buildings Architecture and Engineering*, Ed. F. Mills, 13-22.

[3] LANE-SERFF, G.F., LINDEN, P.F., PARKER, D.J. & SMEED, D.A. 1991 Laboratory modelling of natural ventilation via chimneys. *Architecture and Urban Space*. Eds. S. Alvarez et al. Kluwer Academic Publishers, 505-510.

[4] EDWARDS, M., LINDEN, P.F. & WALKER, R.R. 1994 Theory and practice - natural ventilation modelling. *Proceedings CIBSE Nat. Conference*, **II**, 102-108.

[5] HUNT, G.R. & LINDEN, P.F. 1996 The natural ventilation of an enclosure by the combined effects of buoyancy and wind. Proceedings of ROOMVENT'96, *the Fifth International Conference on Air Distribution in Rooms*, **3**, 239-246.

[6] HUNT, G.R. & LINDEN, P.F. 1997 The fluid mechanics of natural ventilation - displacement ventilation by buoyancy-driven flows assisted by wind. *Building and Environment* (in press).

[7] HUNT, G.R., LINDEN, P.F., KOLOKOTRONI, M. & PERERA, M.D.A.E.S. 1997 Salt bath modelling of air flows, *Building Services Journal, CIBSE*, January 1997, 43-44.

ACKNOWLEDGEMENTS

The authors would like to thank the Building Research Establishment for their financial support of this project through the Department of the Environment's Energy Related Environmental Issues (EnREI) in Buildings programme.

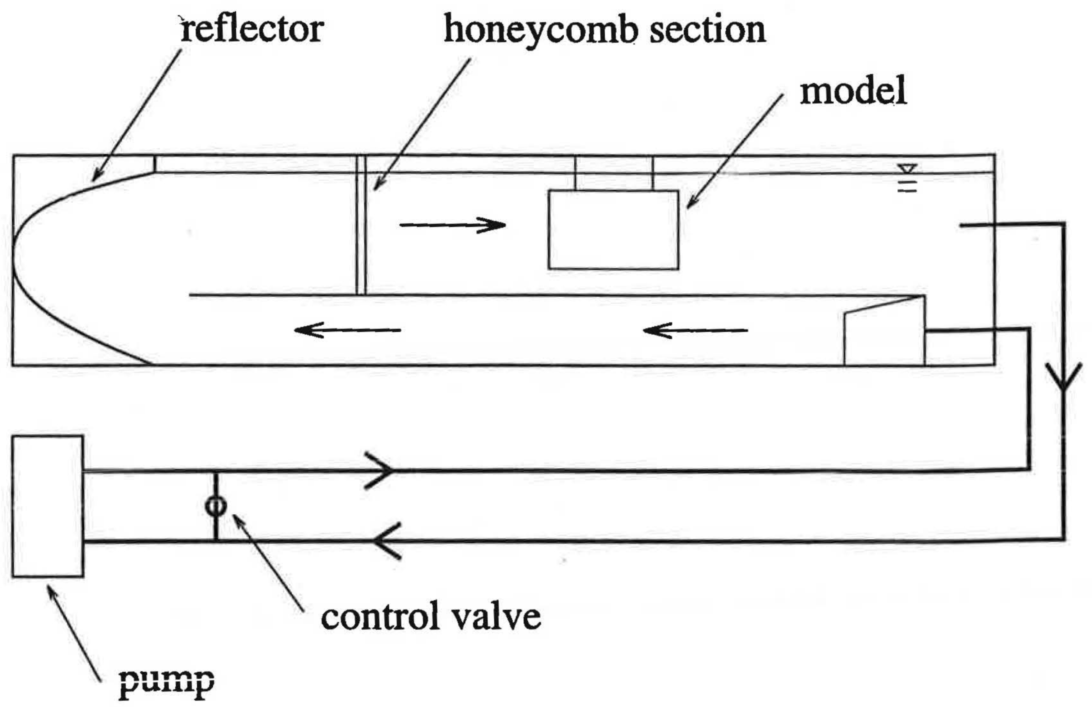

Figure 1. The experimental apparatus.

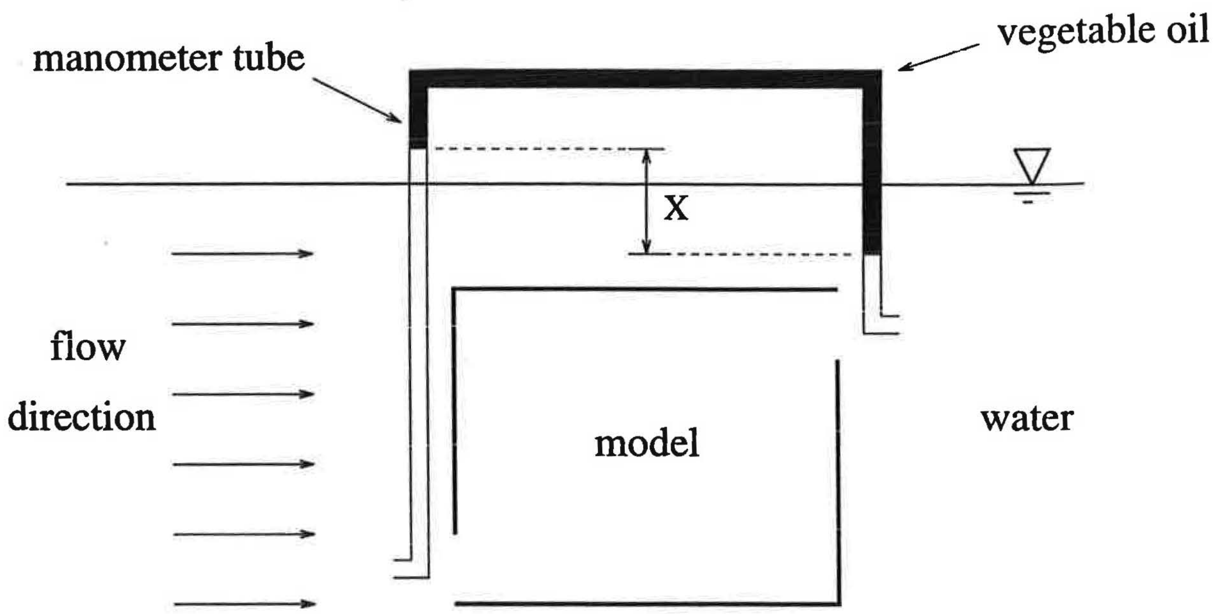

Figure 2. Measuring 'wind' pressure drop using a manometer tube.