5709

Air flow analysis for large spaces in an airport termical building: computational fluid dynamics and reduced-scale physical model tests

R H YAU, BSc, PhD, AMIMechE
Ove Arup and Partners, London, UK.
G E WHITTLE, MSc, PhD, CEng, MIMechE
Arup Research and Development, London, UK.

SYNOPSIS

Computational fluid dynamics (CFD) and reduced-scale physical modelling have been used to optimise air distribution design in a large airport terminal building. The procedures followed are discussed and the role of CFD in the design process is defined.

1 INTRODUCTION

The design of large spaces in buildings has undergone significant changes in the last one or two decades. New types of spaces characterised by atria and other large architecturally dramatic enclosures have evolved. In the United Kingdom, the Lloyds Building, contains an atrium of imposing proportions and style. The Kansai International Airport Terminal Building, which is currently being constructed near Osaka, Japan, contains a very large uncompartmented passenger concourse.

Within the context of innovative buildings, there are increasing demands for ever improved thermal comfort and air quality, and for greater interactivity between building users and their surroundings. Hence, the design of large spaces poses a considerable challenge to architects, services engineers and others concerned with environmental performance and energy use. Major areas of interest encompass air flow, temperature and contaminant distribution, and energy efficiency during both summer and winter operation. Air flow is particularly important because it provides a mechanism for relatively large-scale energy transport processes and has a strong impact on thermal comfort.

2 DESIGN TECHNIQUES FOR PREDICTING THE ENVIRONMENTAL CONDITIONS

2.1 Semi-Empirical Engineering Relationships

Conventional design methods for predicting room conditions and the performance of air distribution systems include semi-empirical engineering relationships to account for particular air distribution methods [1] which is supplemented by previously established engineering design knowledge. However, where non-conventional methods of air supply are proposed or where previous experience is limited, the engineer has to depend on data obtained from other sources, which traditionally have been physical models. Now, with the advent of relatively cheap but powerful computer hardware, another alternative, namely, computational fluid dynamics (CFD) may be used to solve the complex physics of air movement.

2.2 Physical Models

Physical models offer the potential of a 'real world' analogue of a building. But because of size (and hence cost), reduced-scale modelling must be employed where the scale model maintains geometrical similarity with the building but is very much smaller. A model may employ the same 'working fluid' as that of the building, ie. air, or may use another fluid such as water. To ensure that the physical processes occurring in the scale model represent those in the building to an acceptable degree of accuracy, geometrical, hydrodynamic and thermal similarity must be satisfied. The same requirements apply to the imposition of boundary conditions [2]. In practice, it is not always possible to achieve dynamic and thermal similarity concurrently, and within certain ranges of flow conditions it is not necessary. Mullejans [3] carried out non-isothermal tests in mechanically ventilated enclosures at scale factors of 1/1, 1/3 and 1/9 using air as the working fluid where Archimedes number was used as the basis of similarity. The procedure was to operate the models at the same temperature difference and adjust the velocity scale to maintain the Archimedes number. The flow patterns in the three sizes of enclosure were compared and found to agree well, and to be largely independent of Reynolds number. This approach is generally used in air flow reduced-scale physical modelling [7].

2.3 Computational Fluid Dynamics (CFD)

Computational fluid dynamics concerns the representation of the fundamental conservation equations for momentum, energy and mass in mathematical form and their solution to predict fluid flow and convective heat transfer [4,6]. Applied to buildings the method can predict detailed air velocity and temperature fields. The equations to be solved are the conservation equations which are based on the fundamental laws of physics; they are shown in differential form in Figure 1. The momentum and energy equations are known as convection-diffusion equations since they describe how the velocity (in component form) and enthalpy (usually as temperature) is convected with the flow and diffused throughout the calculation domain. To these must be added equations or relationships which define the magnitude of the diffusion characteristic in turbulent flow (a turbulence model). In the k-ε turbulence model [5] this will involve solving additional convection-diffusion equations for the kinetic energy of turbulent fluctuations (k) and its dissipation rate (ε). A model of this type will predict the diffusion coefficient as a field variable rather than as a constant.

In order to solve the differential equations they must first be represented in numerical form. The most common method used is called 'finite volume' which is a form of the finite difference approach. The strong non-linearities demand that an iterative method be used, where an initial estimate of the solution is assumed at the start of the calculation which is improved upon at each iteration. Figure 2 shows an example the finite volume discretised equations in a form ready for solving. At solid boundaries, such as walls, wall function expressions are used to predict shear stress and convective heat transfer coefficient.

3 AIR FLOW ANALYSIS IN AN AIRPORT TERMINAL BUILDING

3.1 Background

A new international airport is being constructed to serve the Kansai region, Japan. Due to noise restrictions and the lack of available land, the airport is being constructed on an artificial island, 5km from the coast, in Osaka Bay. The planning requirements and the free form nature of the roof have defined a number of large spaces. These spaces contain the Arrivals Concourse (Canyon), the International Departures Concourse and the Airside Boarding Lounges (Figure 3).

3.2 Air Conditioning System in the International Departures Concourse

An overall strategy was adopted for the air-conditioning design of the spaces. This was to separate as far as possible the internal occupied areas from the effects of the external environment. Perimeter systems have been designed to offset heat gains and losses at the building envelope, while gains form occupants, lights and machines are offset by local systems. The perimeter zone for the international departure concourse is largely represented by the curved roof surface. To offset heat gains and losses from this fabric, conditioned air is projected from 19 large jet nozzles, spaced at 14.4m centres, beneath the curved roof in the direction from landside to airside. These jets induce a secondary air circulation pattern which ensures that the whole of this `macro' zone is conditioned. With this system, it is important to ensure that the air temperatures and velocities in the occupied areas created by the macro system do not exceed acceptable comfort criteria. The behaviour of the `macro' jets under the various operating conditions are therefore crucial to the success of the system and the space. A specific design evaluation procedure was then devised to study the behaviour of `macro' jets under the various operating conditions.

3.2.1 Semi-Empirical Engineering Relationships

Basic semi-empirical ventilation jet throw equations reported by Baturin [1] were first used to calculate the trajectories of non-isothermal jets into free space in both heating and cooling operating conditions. Further relationships allowed surface attachment effects to be included (Nomura). Both sets of equations indicated that at certain ranges of `macro' jet exit conditions the jets can throw for the required distance which was between 70m to 80m. However, these basic jet flow equations did not take into account the dynamic heat transfer between the jet stream and the ambient environment, and only used a constant Archimedes number throughout the calculation. These were too crude to be used alone for predicting the performance of the buoyant jets into large spaces. Hence, because of the complexity of the flow and the uncertainty of predictions with simple equations, Computational Fluid Dynamics was used to further the evaluations. Several scenarios were studied and the results of these simulations enabled a better understanding of the performance of the jets, which was then used to set up the scale model tests to give more knowledge of the influence of some of the boundary conditions.

3.3 Computational Fluid Dynamics Analysis

In carrying out the CFD analysis three different proprietary CFD codes were used. These were:

(a) PHOENICS (CHAM Ltd.);

(b) Harwell FLOW3D (AEA Technology Ltd.);

(c) STAR-CD (Computational Dynamics Ltd.).

The reasons for the use of three codes were rather complex and represented a mix of considerations, partly historical in that two of the codes were already in-house, and partly to do with a wider requirement to evaluate commercial CFD codes for a number of uses within the Partnership.

The mainly technical considerations regarding selection of the CFD codes for different parts of this analysis depended on a number of factors which included:

(i) the ability to create a curvilinear mesh system to properly represent the curved roof and the initial inclined projection of the large nozzles;

(ii) the need to prescribe the nozzles (mass and momentum sources) within the interior of the calculation domain;

(iii) the availability (and costs) of the code for the computing platforms used in-house;

(iv) a record of successful previous use on building room air movement applications, and particularly the capability to compute buoyant flow.

Analyses were carried out over a period of many months starting at the competition stage prior to work on the project proper.

At this stage PHOENICS was used to demonstrate the possibility of projecting a non-isothermal jet the required distance. At the start of the scheme design, preliminary CFD runs were made using a two-dimensional slice in an attempt to establish the influence of the roof shape on the jet trajectory and the effect of the location of the exhaust position. During these runs it was found that the original position of the exhaust in the Canyon area created unacceptably high velocities on the concourse due to the large bulk movement of air. The exhaust position was therefore re-located to the air-side of the concourse level above head height. These initial runs indicated that the concept of the large nozzles could work.

In order to determine the influence of the macro jet performance on the temperature and air movement in the occupied area, a three-dimensional representative section of one bay was modelled using the Harwell FLOW3D code operated on a SUN 386i workstation. FLOW3D had been identified as a code which had been used successfully in buildings and there was a need to assess its use for wider application in the Partnership. The experience with the implementation was that for the increasing sizes of three-dimensional meshes the turn-around was too slow and cumbersome to meet the tight deadlines for major decisions in the design process. To achieve the response needed at this particular and critical stage of the project a limited number of simulations were commissioned at Harwell on the CRAY-2. This provided the data needed at this stage in the work.

For the final series of simulations, prior to physical model testing, the STAR-CD code was used. This code was already available in-house on a VAX 11/785 and had recently been updated with the ability to specify mass and momentum sources within the domain interior. The updated code offered particular flexibility in generating complex three-dimensional meshes and post-processing results. The VAX 11/785 mini-computer ran slightly faster than the SUN 386i workstation by a factor of approximately 1.5, and had a much larger main memory.

Subsequent decisions were made to include only the international departures concourse in the STAR-CD simulations since the main interest was in the relative performance of the macro jets. The incorporation of the Canyon space into the simulation would have resulted in longer run times and possibly an inability to meet the design programme schedule. The three-dimensional body-fitted co-ordinate mesh contained approximately 16,000 cells which is rather coarse particularly by todays standards (Figure 4). A much finer 3D mesh was not pursued because it would be beyond the time constraint and the capability of the computer hardware (VAX 11/785). Recent in-house experience has been gained using a CONVEX C210, which, as expected, dramatically reduces iteration times and allows the building of much more detailed and complex meshes.

The results of different scenarios are:

(1) In an isothermal simulation, the jet of macro air attaches to the roof until it passes the macro exhaust. The circulation which is induced creates average air velocities of 0.28m/s in the occupied zone (Figure 5).

(2) In summer simulation 'A', the cold macro jet detaches from the roof and

reaches the floor at approximately 30m from the landside of the main concourse. This early detachment of jet due to strong buoyancy forces (characterised by high Archimedes number) creates a strong draught at occupancy level which could result in discomfort (Figure 6).

(3) In summer simulation 'B', the cold macro jet remains attached to the roof because of reduced buoyancy forces. However, because of buoyancy the jet gradually thickens (downwards) as it projects towards the air-side of the main concourse. The air velocities and air temperatures at occupancy level remain within acceptable comfort criteria (Figure 7).

(4) In summer simulation 'C', the cold macro jet stays attached to the roof as a result of the lower Archimedes number following from increased jet momentum. Higher air velocities are induced in the occupied zone although they are still within the comfort criteria (Figure 8).

3.4 Reduced-Scale Physical Model Tests

The CFD simulations were used to study air movement patterns and temperature distributions within the space at a number of design conditions and to demonstrate that the system has the potential to work as envisaged. The information generated was then used to define a programme for the physical model tests. The physical tests, which involved measurements in a 1/10 scale model (which was still large enough to walk within), were undertaken to provide confirmatory evidence of the acceptability of the air distribution design and to give the opportunity to optimise performance.

In order to keep the size of the model within certain constraints the underlying modularity of the building was identified based on a section comprising typical internal structure and air handling systems. This contained the check-in counter, airside WC block and Macro supply nozzles. The model constructed, which was to 1/10 scale, comprised two structural bays of the full cross-section of the concourse level and canyon of the passenger terminal building. It was constructed and tested at the Building Services Research and Information Association (BSRIA). The model was designed to simulate as closely as possible the actual conditions experienced during the design summer and winter loads. Fabric heat gains and losses, internal heat gains and the micro air systems were simulated. The main space was conditioned by the `macro' system air supply nozzles, set to discharge at an inclined angle upwards across the ceiling surface. It was the angle of discharge, the supply air volume flow rate and discharge velocity which were to be optimised in the investigation of thermal comfort conditions.

To better model heat transfer through the roof, a temperature controlled void was constructed above the insulated ceiling. The roof insulation was represented by glass fibre mat above the hardboard ceiling construction. Within the ceiling-void electric heater tapes and controller were used to simulate the thermal condition generated by a combination of outdoor temperature and solar gains.

Figure 9 shows an interior view of the physical model.

The testing was performed in two stages:

(1) Smoke Movement Patterns

Smoke was injected into the jet air stream and a visual record was made of its movement across the space noting the flow pattern, point of detachment from the roof and direction of flow on the concourse. Patterns were established for a range of temperature conditions, mass flow rates and velocities, and angles of jet trajectory relative to the roof (Figure 9). The smoke testing was compared with the CFD results and to determine the optimum conditions to carry out the detail measurement phase.

Figure 10 indicates the general jet projections identified from the smoke tests in the physical model.

(2) Detailed Measurements

Detailed measurements of velocity and temperature were made on a regular grid throughout the space. These measurements were plotted to established air velocity and temperature profiles and used to calculate comfort indices for the occupied zone.

3.5 Comparing Results of CFD and Physical Model Tests

The behaviour of the ventilation jets under different summer operating conditions predicted by the CFD model were generally confirmed by the reduced-scale physical model tests. Attempts to compare the exact numeric values from both tests were not made because the CFD model, which was used as first-stage design tool, was not set up for such comparison. However, the CFD and physical model tests proved to be complementary in evaluating the thermal environment and were valuable in verifying the concept of removing the heat from the roof surface by the macro jet, and the effectiveness of this novel air distribution system. Although not intended at the time, it is recognised that much more could be done to compare measured and predicted velocity and temperature profiles and to add to the establishing of confidence levels in the use of CFD.

4 CONCLUSION

A description is presented of the use of CFD and a reduced-scale physical model to develop the room air distribution design system for a large airport passenger terminal building. It is shown that CFD provides a valuable tool in the design process to complement, in this particular project, more traditional physical model tests.

5 REFERENCES

1 BATURIN, V. V. (1972), Fundamentals of industrial ventilation. Pergamon Press, Oxford.

2 PARCZEWSKI, K. I. and RENZI, P. N. (1963), Scale model studies of temperature distributions in internally heated enclosures. ASHRAE Trans., Vol 69, pp453-463.

3 MULLEJANS, H. (1966), The similarity between non-isothermal flow and heat transfer in mechanically ventilated room. Westdentscher Verlag, Koln & Opladen.

4 PATANKAR, S. V. (1980), Numerical heat transfer and fluid flow. McGraw Hill Pub., USA.

5 LAUNDER, B.E. and SPALDING, D.B. (1974), The numerical computation of turbulent flows, Computer methods in applied mechanics and engineering, Vol 3.

6 WHITTLE, G.E. (1986), Computation of air movement and convective heat transfer within buildings, Int. J. of Ambient Energy, Ambient Press Ltd.

7 WHITTLE, G.E. (1990), Air flow modelling in atria. Proceedings of IMechE Symposium on Atrium Engineering, IMechE, London.

Momentum Equation in X direction

$$\underbrace{\frac{\partial}{\partial t}(\rho U)}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial x}(\rho UU) + \frac{\partial}{\partial y}(\rho VU) + \frac{\partial}{\partial z}(\rho WU)}_{\text{convection}} = \underbrace{-\frac{\partial P}{\partial X}}_{\text{pressure gradient}}$$

$$+ \underbrace{\frac{\partial}{\partial X}\left(\mu_{eff}\frac{\partial U}{\partial X}\right) + \frac{\partial}{\partial Y}\left(\mu_{eff}\frac{\partial U}{\partial y}\right) + \frac{\partial}{\partial Z}\left(\mu_{eff}\frac{\partial U}{\partial Z}\right)}_{\text{diffusion}}$$

$$+ \underbrace{(\rho-\rho_{\infty})\,g_x}_{\text{buoyancy}} + \underbrace{S_u}_{\text{momentum source or sink}}$$

Similar equations to the above exist for the Y and Z directions.

where:

C_p	=	specific heat
g_x	=	gravitation acceleration in X-direction
h	=	enthalphy (C_p.T)
P	=	pressure
S_u, S_h	=	sources (sinks) of momentum, energy
t	=	time
T	=	temperature
U, V, W	=	velocity components in X,Y,Z directions
X, Y, Z	=	co-ordinate directions
ρ, ρ_{∞}	=	density and reference density
μ_{eff}, Γ_{eff}	=	diffusion coefficients for momentum, energy.

Mass Conservation Equation

$$\underbrace{\frac{\partial \rho}{\partial t}}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial X}(\rho U) + \frac{\partial}{\partial Y}(\rho V) + \frac{\partial}{\partial Z}(\rho W)}_{\text{convection}} = 0$$

Energy Conservation Equation

$$\underbrace{\frac{\partial}{\partial t}(\rho h)}_{\text{transient term}} + \underbrace{\frac{\partial}{\partial X}(\rho Uh) + \frac{\partial}{\partial Y}(\rho Vh) + \frac{\partial}{\partial Z}(\rho Wh)}_{\text{convection}} =$$

$$\underbrace{\frac{\partial}{\partial X}\left(\Gamma_{eff}\frac{\partial h}{\partial X}\right) + \frac{\partial}{\partial Y}\left(\Gamma_{eff}\frac{\partial h}{\partial Y}\right) + \frac{\partial}{\partial Z}\left(\Gamma_{eff}\frac{\partial h}{\partial Z}\right)}_{\text{diffusion}} + \underbrace{S_h}_{\text{energy source or sink}}$$

Fig 1 The conservation equations

Discretised mass continuity equation

$$(\rho_P - \rho_P^{o}).\mathrm{Vol} / dt + C_N + C_S + C_E + C_W + C_H + C_L + b_c = 0$$

where b_c = mass source

Discretised energy equation

$$a_P h_P = a_N h_N + a_S h_S + a_E h_E + a_W h_W + a_H h_H + a_L h_L + b_h$$

where h = enthalpy (C_pT)
Cp = specific heat
T = temperature

b_h = energy source + $\mathrm{Vol}.\rho_P^{o}.h_P^{o} / dt$

Discretised momentum equation in X direction

$$a_P U_P = a_N U_N + a_S U_S + a_E U_E + a_W U_W + a_H U_H + a_L U_L + b_U$$

where a = convection / diffusion fluxes
- a function of (C, D)
C = convection (mass) flux
D = diffusion flux

$a_P = a_N + a_S + a_E + a_W + a_H + a_L + \mathrm{Vol}.\rho_P / dt$

b_U = momentum source + $\mathrm{Vol}.\rho_P^{o}.U_P^{o} / dt$

Vol = cell volume
ρ = density
U = computed velocity
U^{o} = velocity at previous time step
dt = time step

Subscript P refers to the in-cell variable value, and subscripts N, S, E, W, H, and L refer to surrounding north, south, east, west, high and low variable values.

Superscript 'o' refers to the variable value at the previous time step.

The form of the function defining a_N, etc, follows from the selection of the discretisation scheme, eg. upwind, central, hybrid differencing.

Fig 2 The discretised conservation equations

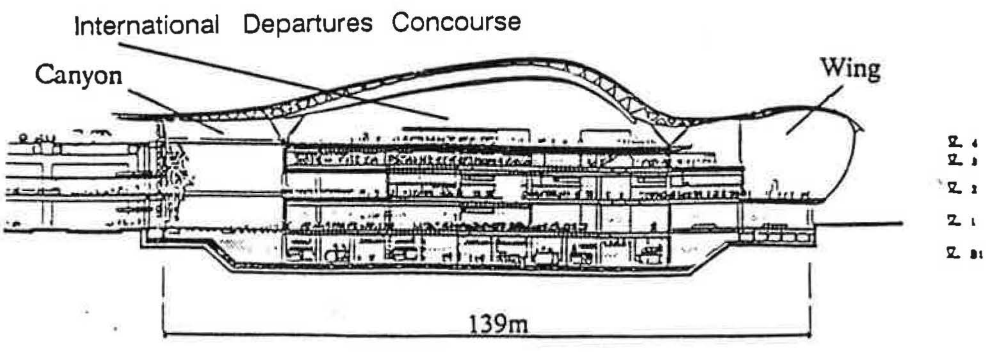

Fig 3 Cross-section of terminal building

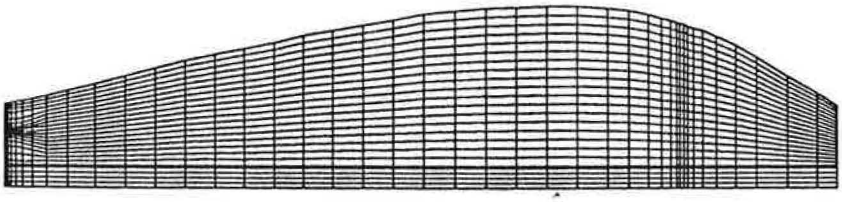

Fig 4 Curvilinear grid mesh for CFD analysis

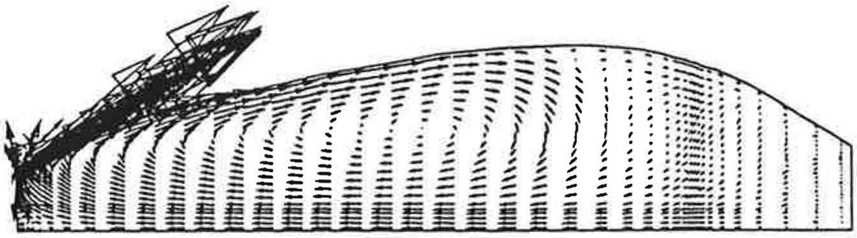

Fig 5 Isothermal simulation

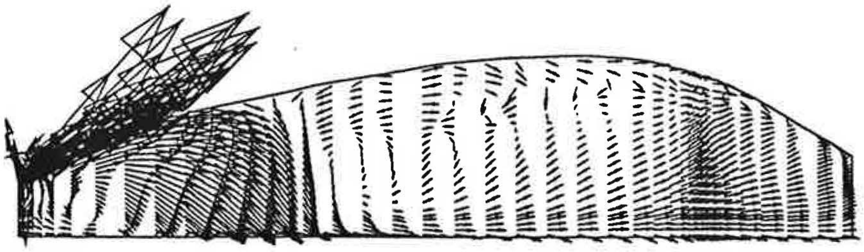

Fig 6 Summer simulation (A)

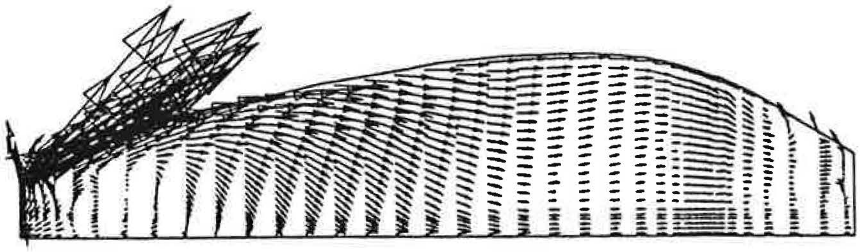

Fig 7 Summer simulation (B)

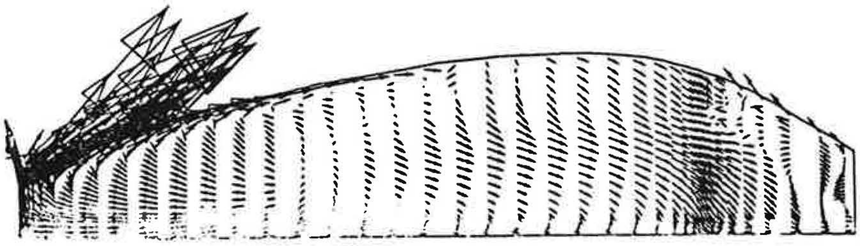

Fig 8 Summer simulation (C)

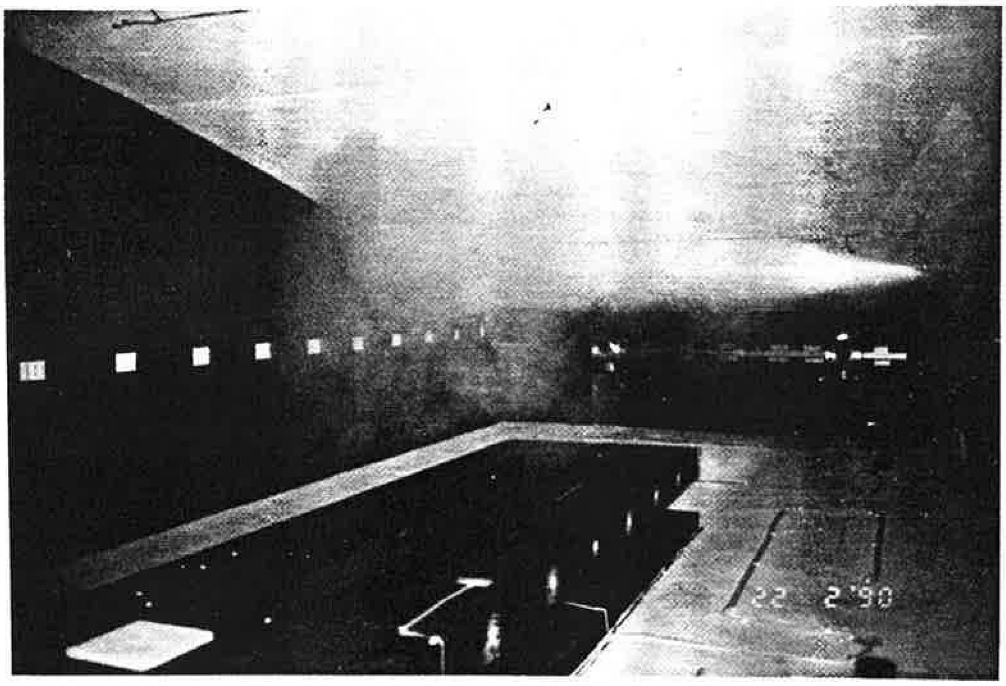

Fig 9 Physical model test for Kansai International Airport Terminal

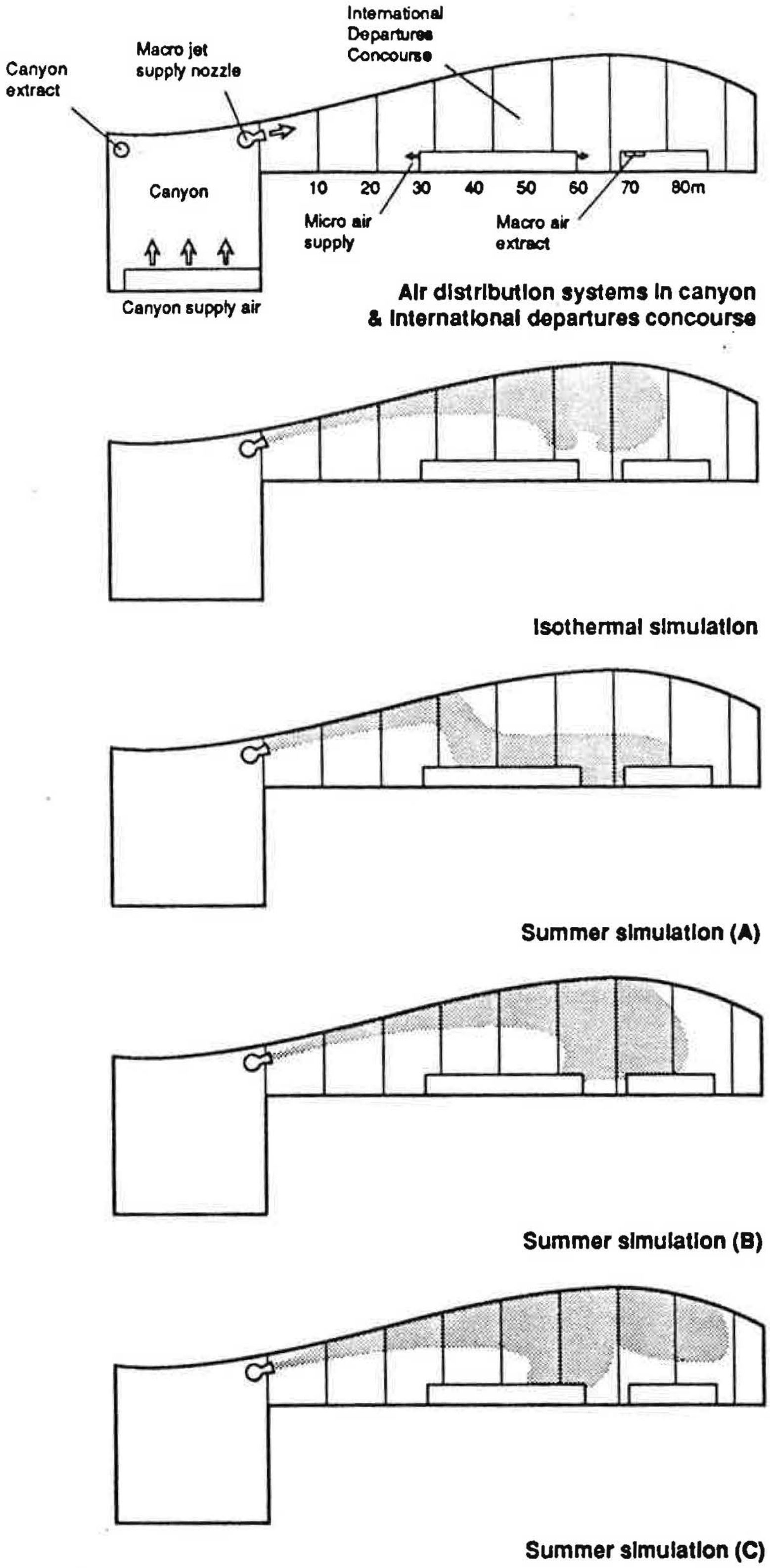

Fig 10 Smoke tests in reduced scale model