#608

International Symposium on Room Air Convection and Ventilation Effectiveness University of Tokyo, July 22 - 24, 1992

AIR FLOW STUDIES IN LARGER SPACES: A CASE STUDY OF KANSAI INTERNATIONAL AIRPORT PASSENGER TERMINAL BUILDING, OSAKA, JAPAN

A Guthrie B.Eng., Msc., C.Eng., MCIBSE, MASHRAE Ove Arup

Hualdity (%)

h

H Ikezawa AIJ, SHASE, Accredited Consulting Engineer Nikken Sekkei K Otaka AIJ, SHASE, Accredited Consulting Engineer Nikken Sekkei



R M H Yau BSc., Ph.D., C.Eng.,MIMechE Ove Arup

ABSTRACT

This paper describes a case study into the methods used to predict air movements, temperatures and comfort indices for a large space. Use of CFD techniques and physical modelling are discussed.

KEYWORDS Air Distribution, CFD, Modelling

INTRODUCTION

There has always been an architectural need to create large internal spaces.These range from Cathedrals to Rocket Sheds and from Sports Halls to Railway Stations. In the past the environmental performance of these large spaces has either been self-regulating or controlled by experience. New construction techniques and a desire for better comfort have forced the Engineer into the need to predict the environmental performance of large Buildings.

An international architectural competition was held in 1988 for the design of the passenger terminal building for the new Kansai airport. The design for the building involves two major zones; the Main Terminal Building (MTB) which houses the check-in, immigration and baggage handling activities; and the Wings which accommodate the departure lounges and aircraft boarding gates. The total length of the building 'from wing-tip to wing-tip' is just over a mile, and will accommodate 48 berthed aircraft. An automatic guideway transit system is used to carry passengers out along the Wings to and from the MTB.

DESIGN PHILOSOPHY

The air-conditioning concept for the large International Departure Lounge was to provide as much comfort conditioning as possible using large air jets situated at one side of the space. The curve of the roof from landside to airside was designed to facilitate the trajectory of the jet and curved ceiling panels are hung under the roof to provide a channel for the air to blow against. These ceiling

M

de

of

CL

th

in

a

C

Ps

a

e

F

i.t

panels also act as reflectors for the post-mounted uplighters and an architectural finish to the ceiling. Long throw jets were used for several reasons.

1. They eliminate air supply ductwork from the roof or floor of the space, 2. They remove heat flow through the glass areas of the roof at source, 3. They provide air movement for comfort within the space, 4. They provide outside air to the space.

To supplement the cooling provided by the general 'Macro' system, recirculating 'Micro' systems are used in high occupancy areas such as around check-in desks, to provide local cooling. Floor mounted, fan coil posts, blowing air at the glass provide additional heating and cooling at the perimeter.

In order to determine the velocity temperature and angle of discharge of the large jets and their effectiveness in removing high level heat and providing ventilation to the space a clear understanding of the airflow patterns and temperature distributions was r.ecessary. There were two other important questions that needed early consideration. Firstly, the effect of the 30m high entrance area, referred to as the canyon, on the air flow patterns and secondly, the effect of the location of the extract grilles on the air distribution.

Both Computational Fluid Dynamics and scale model testing were used to resolve these design questions, and to predict comfort indices for the space.

COMPUTATIONAL FLUID DYNAMICS

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. Applied to buildings the method can predict detailed air velocity and temperature fields. The equations are based on the fundamental laws of physics; they are shown in differential form in (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, equations or relationships which define the magnitude of the diffusion characteristic in turbulent flow (a turbulence model) must be added. In the k- ε turbulence model (2) this involves solving additional convection-diffusion equations for the kinetic energy of turbulent fluctuations (k) and its dissipation rate (ε). A model of this type predicts the diffusion coefficient as a field variable rather than as a constant.

In order to solve the differential equations they must 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. At solid boundaries, such as walls, wall function expressions are used to predict shear stress and convective heat transfer coefficient.

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, i.e. air, or may use another fluid such as water. To ensure that the physical processes occurring in the scale model accurately represent those in the building geometrical, hydrodynamic and thermal similarity must be satisfied. The same requirements apply to the imposition of boundary conditions (3). 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 (4) 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 patters in the three sizes of enclosure were compared and found to agree well, and to be largely independent of Reynold's number. This approach is generally used in air flow reduced-scale physical modelling (5).

COMPUTATIONAL FLUID DYNAMICS ANALYSIS

In carrying out the CFD analysis, three different proprietary CFD codes and a separate research CFD code were used in the UK and in Japan respectively.

These were:

r

e

ns

re

st

а

he

!d

у 1g, (a). PHOENICS (CHAM LIMITED UK), (b). Harwell FLOW3D (AEA Technology Limited, UK), (c). STAR-CD (Computational Dynamics Limited, UK), (d). CFD code (Murakami and Kato Laboratory, the University of Tokyo, Japan).

The reasons for the use of three codes in the UK 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 requirement to evaluate CFD codes which best suit the problem. A separate CFD exercise was undertaken simultaneously by Murakami and Kato Laboratory of the University of Tokyo at the early stage of the design. This simulation, included the evaluation of the initial design condition.

The main technical criteria for 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 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. At this stage PHOENICS was used to demonstrate the possibility of projecting 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 exhaust location. During these runs it was found that the original position of the exhaust in the Canyon area created unacceptably high velocities on

concourse level and canyon of the passenger terminal building. 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 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 surfaces. To 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.

th

re

ru

te

re

ac

lin

CI

be

CC

de

ST

m

ap

it p

ve

rea

Th

Ares

be

gr: co

ac

at

Hi

CO

RE

pr 1/

cc th

ur

th

CC

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. It was recognised that winter conditions would not pose the same difficulties for the Macro supply jet that the summer conditions would, therefore summer operating regimes were examined in greater detail. Figure 7 indicates the general jet projections identified from the smoke tests in the physical model.

Under isothermal conditions the optimum angle of discharge for the jets was 30 degrees. Various summer regimes were examined and the optimum angle of discharge for each condition were found to lie between angles of 25 degrees and 27.5 degrees.

The smoke testing was compared with the CFD results and to determine the optimum conditions to carry out the detail measurement phase.

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

For the maximum occupancy summer operating condition the mean predicted percentage dissatisfied (PPD) for the occupancy zone is 14.3% (PMV: 0.6). Values of PPD were found to range from 7.2% (PMV: 0.33) to 21.7% (PMV: 0.26). Examination of the predicted mean votes (PMVs) shows this discomfort tends to be at the warm end of the scale and not the cool end. Some of the regions in the concourse under maximum design occupancy experienced slight warm discomfort but this was considered acceptable.

For the winter operating condition the mean PPD for the occupancy zone is 5.2% (PMV:0.11). Values of PPD ranged from 5.0% (PMV: 0.02) to 6.4% (PMV: 0.26). Most of the data provides for slightly warm discomfort in the space, though some areas do experience very slight cold discomfort. Conditions under the winter regime would prove acceptable to at least 90% of the occupants.

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

the concourse due to the large bulk movement of air. The exhaust was therefore relocated to the air-side of the concourse level above head height. These initial runs indicated that the concept of the large nozzles could work.

зi

d

hat

at ed

as

1

he

is

٦

er

ł

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. To achieve the response needed at this particular and critical stage of the project a limited number of simulations were commissioned on a super computer.

For the final series of simulations, prior to physical model testing, the STAR-CD code was used. This code was already available in-house and had recently been updated with the ability to specify mass and momentum sources within the interior domain. The updated code offered particular flexibility in generating complex three-dimensional meshes and post-processing results. 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 three-dimensional body-fitted co-ordinate mesh contained approximately 16,000 cells. This is rather coarse by todays standards (Figure 2).

The results of different scenarios described in Table 1 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 3).

(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 4).

(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 velocities and air temperatures at occupancy level remain within acceptable comfort criteria (Figure 5).

(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 velocities are induced in the occupied zone although they are still within the comfort criteria (Figure 6).

REDUCED-SCALE PHYSICAL MODEL TESTS

The information generated by the CFD simulations was used to define a programme for the physical model tests. The physical tests were undertaken on a 1/10 scale model (which was still large enough to work within), 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 the check-in counter, airside WC block and Macro supply nozzles. The model constructed, comprised two structural bays of the full-cross section of the 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.

12 8m

Canyo

CONCLUSION

CFD and physical modelling provide useful and complimentary tools in evaluating complex airflow patters and temperature distribution in large spaces.

REFERENCES

1. PATANKAR, S.V. (1980), Numerical heat transfer and fluid flow. McGraw Hill Pub., USA; 2, LAUNDER, B.E. and SPALDING, D.B. (1974), The numerical computation of turbulent flows, Computer methods in applied mechanics and engineering, Vol 3; 3. PARCZWESKI, K.I. and RENZI, P.N. (1963), Scale model studies of temperature distributions in internally heated enclosures. Ashrae Trans., Vol 69, pp453-463; 4. MULLEJANS, H. (1966), The similarity between nonisothermal flow and heat transfer in mechanically ventilated room, Westdeutscher Verlag, Koln & Opladen; 5. WHITTLE, G.E. (1990), Air flow modellings in Atria. Procoodings of IMechE Symposiumon Atrium Engineering, IMechE, London.

Toat	Type of Simulation	Jet Exit Velocity (m/s)	Nozzle Inclination Angle	Jet Exit Temparature (C*)	Temparature difference between Jet ∡T (°C)
_1	Isothermal	6.0	30°	14 M	
2	Summer (a)	6.0	30°	16.0	10.0
3	Summer (b)	6.0	30	21.0	5.0
4	Summer (c)	9.4	30°	19.87	6.13
5	Winter	6.0	30	22.4	-2.4







Cross section of terminal Building



З

12.8m

80m