AIR MOVEMENT & VENTILATION CONTROL WITHIN BUILDINGS

12th AIVC Conference, Ottawa, Canada
24-27 September, 1991

PAPER 3

MODELS FOR THE PREDICTION OF ROOM AIR DISTRIBUTION

PETER V. NIELSEN

University of Aalborg
Sohngårdsholmsvej 57, DK-9000 Aalborg, Denmark

SYNOPSIS

The paper describes work on simplified design methods made in connection with the International Energy Agency programme "Air Flow Pattern within Buildings", Annex 20, subtask 1.

It is shown that simplified models are able to indicate design values as the maximum velocity in the occupied zone and penetration depth of a non-isothermal jet in a room.

The design according to throw of an isothermal jet is a fully developed method which has a sufficient level of accuracy when it is used in regular rooms. Models for prediction of the maximum velocity in the occupied zone and penetration depth of non-isothermal jets need further development.

The possibility of Computational Fluid Dynamics (CFD) is evaluated and it is compared with simplified models. It is shown that the CFD-method is special useful because it gives the distribution of the variables as well as the design values. The CFD-method can also predict variables which are time consuming to measure by full-scale experiments.

The CFD-method is especially useful for the prediction of air distribution in large enclosures with complicated geometry and different sources for the air movement.

LIST OF SYMBOLS

a_o	Supply area	m^2
Ar	Archimedes' number	
c_p	Specific heat	J/kg°C
g	Gravitational acceleration	m/sec^2
H	Height of room	m
K_a	Constant for diffuser	
K_{sa}	Constant for calculation of penetration depth	
L	Length of room	m
ℓ_{Th}	Throw	m
n	Air change rate	h^{-1}
q_o	Flow rate	m^2/s
T_o	Supply temperature	°C
T_R	Return temperature	°C
u_L	Velocity in wall jet with lenth L	m/s
u_o	Supply velocity	m/s
u_{rm}	Maximum velocity in occupied zone	m/s
u_{Th}	Reference velocity	m/s
u_x	Velocity in wall jet with length x	m/s
W	Width of room	m
x	Length of wall jet	m
x_o	Distance to virtual origin	m
x_s	Penetration depth of wall jet	m

β	Volume expansion coefficient	K^{-1}
ε	Ventilation efficiency	
ρ_o	Density	kg/m^3

1. INTRODUCTION

The International Energy Agency (IEA) has the basic aim to promote co-operation among the twenty-one participating countries, to increase energy security through energy conservation, development of alternative energy sources and energy research development and demonstration. One programme in this activity is "Energy Conservation in Building and Community Systems".

Annex 20, subtask 1, in this programme works with predictions of accurate air flow patterns within rooms in order to obtain maximum energy efficiency, high thermal comfort and optimum indoor air quality.

The purpose of the project is to formulate requirements for design tools and to lay the groundwork for their development. Computational Fluid Dynamics (CFD) is one of the main activities in subtask 1, but experimental work in full-scale rooms does also play an important role.

It is also desirable that some attention should be devoted to simplified design methods. This paper is a part of this decision. It discusses the limitation and possibility of the simplified models in comparison with CFD-codes and shows the necessary boundary conditions needed for the models.

The evaluation of the simplified design models is supported by comparisons with measurements made in test rooms in different countries.

The following models are discussed

- design according to throw of isothermal jet
- maximum velocity in occupied zone
- penetration depth of non-isothermal jet

and for comparisons

- prediction of air distribution with a CFD-code.

The subtask 1 work involves development work on other simplified models as e.g. zonal models and data base models but those activities are not discussed in this paper.

2. DESIGN ACCORDING TO THROW OF ISOTHERMAL JET

The design models described in this paper are all based on the theory of self-similar jet flow. Figure 1 shows an example of this type of flow in a room with sidewall mounted grille. The flow below the ceiling is a self-similar wall jet which is rather independent of the downstream room geometry, which means that it is independent of room height and room length.

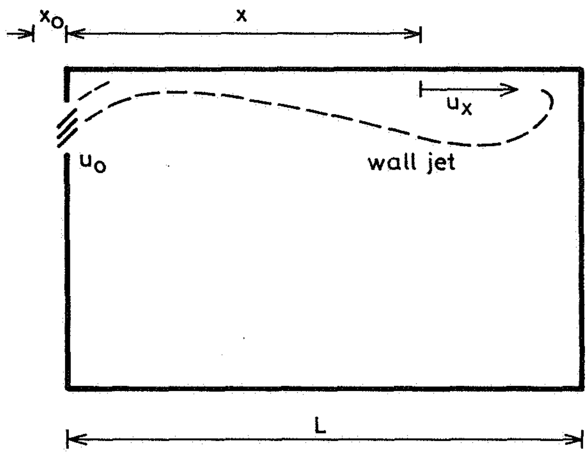

Figure 1. Wall jet in a ventilated room.

The velocity decay in the three-dimensional wall jet below the ceiling is given by:

$$\frac{u_x}{u_o} = K_a \frac{\sqrt{a_o}}{x + x_o} \tag{1}$$

where u_o and u_x are supply velocity and maximum velocity in the wall jet in the distance x from the opening, respectively. a_o is the supply area of the diffuser and x_o is the distance to the virtual origin of the wall jet. K_a is a constant.

K_a takes values from 2 to 10 and x_o is about zero, dependent on the actual diffuser and diffuser location. K_a, a_o and x_o may be dependent on the Reynolds number in the case of low turbulent flow.

A length ℓ_{Th}, called the throw, is defined as distance from the opening to a location where the maximum velocity u_x is equal to a given reference value u_{Th}, see reference [1].

$$\ell_{Th} = \frac{u_o K_a \sqrt{a_o}}{u_{Th}} - x_o \qquad \text{(m)} \tag{2}$$

It is the purpose of the design procedure to control the air distribution in the room in such a way that the maximum velocity in the occupied zone u_{rm} is up to 0.15 m/s. General experience shows that this is achieved when the throw ℓ_{Th} is equal to room length L and the reference velocity u_{Th} is equal to 0.2 m/s or 0.25 m/s.

Producers of Air Terminal Devices are presenting equation (2) as design charts where it is possible to find the flow rate q_o when the diffuser (x_o, a_o and K_a) and the throw ℓ_{Th} are selected. The throw is normally recommended to be the room length in the situation in figure 1. More generally the throw is the half lenght between two diffusers with opposite position or the length between diffuser and wall. Other definitions of ℓ_{Th} may be used to compensate for different diffuser designs and different room geometry. A throw of $L + H$ - 1.8 m is for example used when the room is high and it expresses formally that the maximum velocity in the wall jet is supposed to be equal to u_{Th} when it passes through the occupied zone.

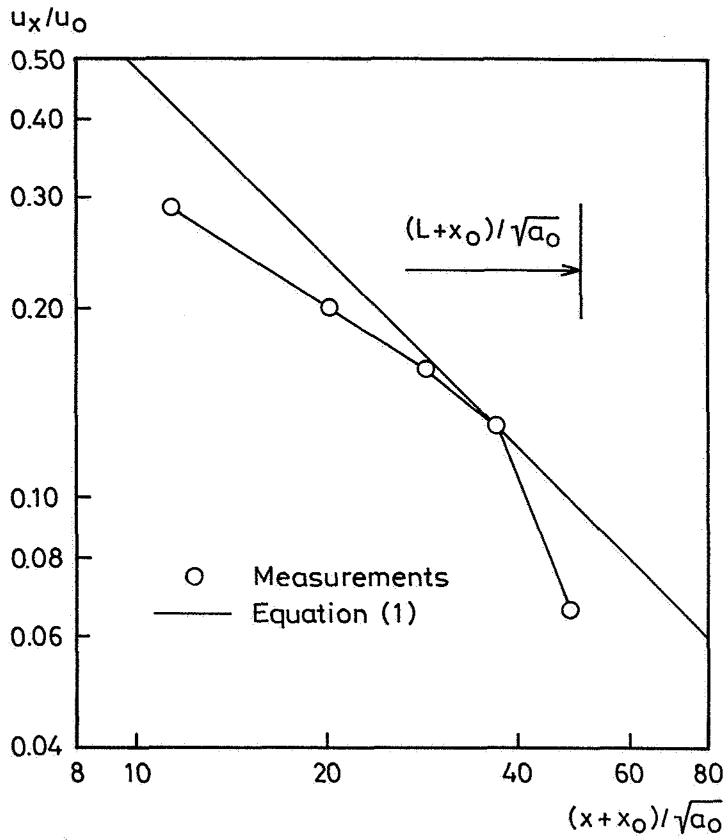

Figure 2. Measurement of maximum velocity in the wall jet as a function of distance, see reference [2]. Air change rate 3h^{-1}. Room and diffuser are specified according to the requirements in references [3] and [4].

The design of supply conditions in the IEA test room, according to throw of isothermal jet, is based on measurements of the velocity decay shown i figure 2. The line for equation (1) corresponds to a K_a-value equal to 4.8. This is a value which ensures that calculated velocities in the wall jet always are higher or equal to the measured values, and this is a normally accepted procedure to ensure that too high velocities in the occupied zone are avoided.

Equation (2) shows that the design supply veloicty u_o is equal to 2.09 m/s under the following assumptions

$$a_o = 0.00855 \text{ m}^2, \quad K_a = 4.8 \text{ m}, \quad x_o = 0.45 \text{ m}$$

$$\ell_{Th} = 4.2 \text{ m} \quad \text{and} \quad u_{Th} = 0.2 \text{ m/s}$$

The design supply velocity corresponds to a specific air flow rate of 1.78h^{-1} in the test room.

Figure 3 shows the measured level of maximum velocity in the occupied zone as well as the predicted values with CFD-codes, see reference [5]. The maximum velocity in the occupied zone will have the following values

$\sim$ 0.08 m/s (Design according to throw of isothermal jet)

$\sim$ 0.08 m/s (CFD-code, Basic case)

$\sim$ 0.12 m/s (CFD-code, Prescribed Velocity Method)

which should be compared to an expected value of 0.15 m/s.

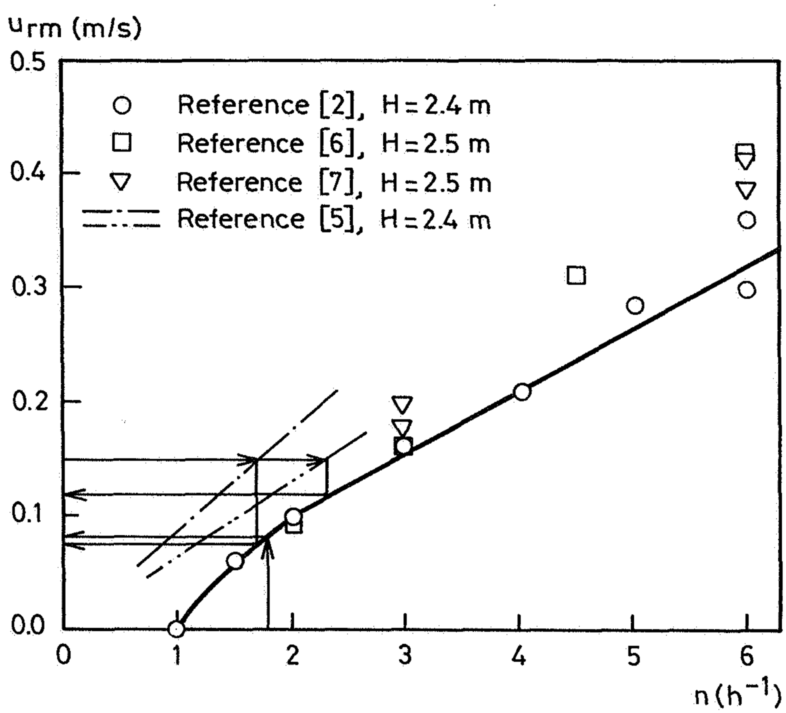

Figure 3. Maximum velocity in the occupied zone. Measurements and predictions from CFD-codes.

There is some deviation in the measurement shown in figure 3 which should be considered when the different design methods are evaluated. This situation is also expressed in figure 4 which shows the measurement of velocity decay in the isothermal wall jet made by the countries, Denmark [2], Sweden [6], Finland [7], and Germany [8].

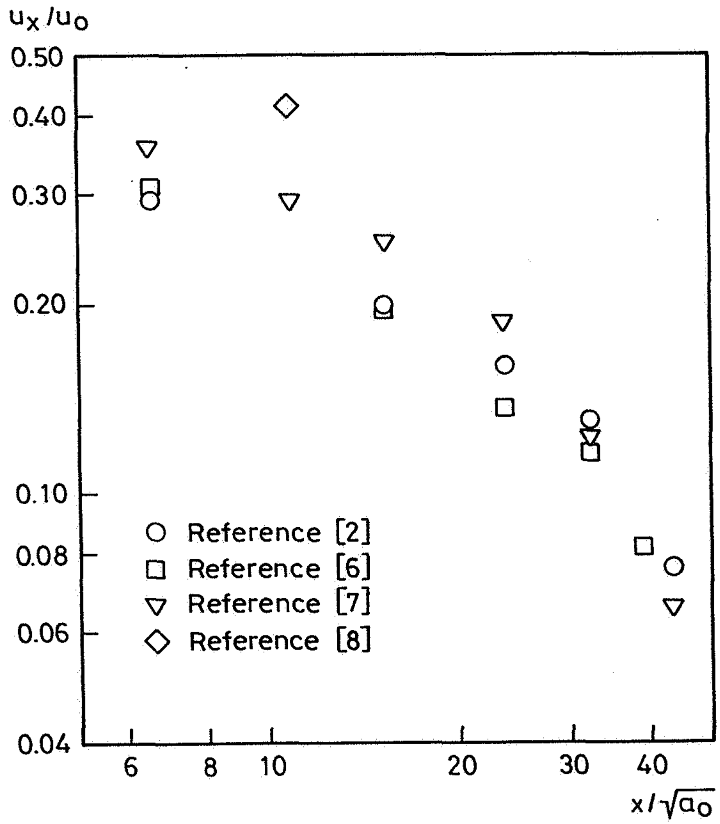

Figure 4. Velocity decay versus distance measured in four different countries. Test case B2.

Furthermore, it should be considered that the actual velocities in the occupied zone may deviate from design velocities because it is difficult to adjust every single diffuser, and because there are small differences in the diffusers. The velocity level is also influenced by individual furnishings in the rooms.

It may be concluded that all the design methods are able to predict a maximum velocity in the occupied zone which is comparable with measured values.

It is possible to make further improvement in both the design, according to throw of isothermal jet, and in the design based on a CFD-method.

3. MAXIMUM VELOCITY IN THE OCCUPIED ZONE

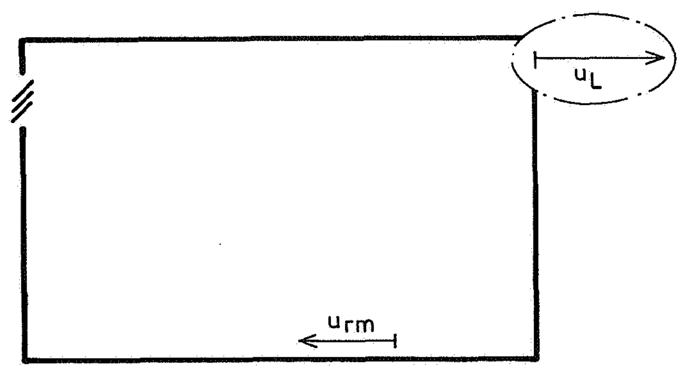

Figure 5. Location of the reference velocity u_L and location of the maximum velocity u_{rm} in the occupied zone.

The maximum velocity in the reverse flow u_{rm} is located close to the floor at a distance of $\sim 2/3\ L$ from the supply opening. This velocity is also the maximum velocity in the occupied zone in cases where the jet below the ceiling and the jet at the end wall are outside the occupied zone. Experiments with isothermal flow show that u_{rm} is a simple function of a reference velocity u_L, which is the velocity in an undisturbed wall jet of the length L from the actual diffuser, see Hestad [9]. The velocity u_L contains information on supply velocity, distance from inlet and geometrical details around the initial flow, such as type of Air Terminal Device, adjustable blades and distance from ceiling. The relation between u_{rm} and u_L, as well as the level of u_L, can therefore be used as elements in a design procedure.

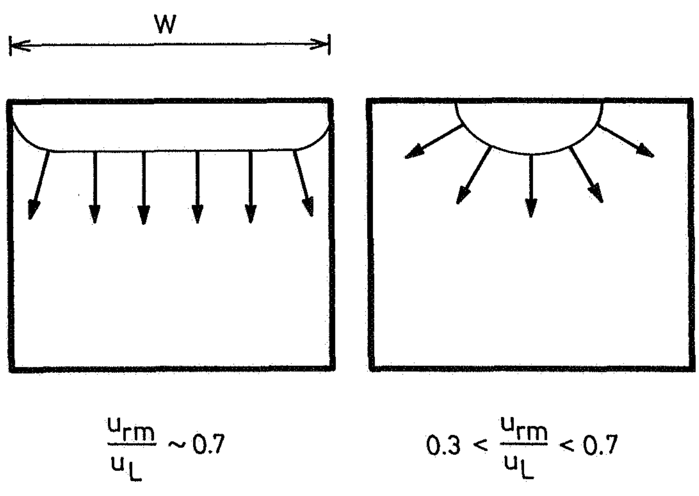

Figure 6. Flow at the end wall opposite the supply opening in case of two-dimensional and three-dimensional air movement.

The sketch in the left side of figure 6 shows the flow at the wall opposite the supply opening in case of a two-dimensional flow. The deflected jet flows vertical down the end wall as a new two-dimensional wall jet, and experiments show that u_{rm}/u_L is egual to 0.7, see reference [10]. A design according to throw of isothermal jet will in this case give a maximum velocity in the occupied zone of $u_{rm} = 0.14$ m/s ($\ell_{Th} = L$, $u_{Th} = 0.2$ m/s).

The sketch in the right side of figure 6 shows the air movement at the end wall in case of a three-dimensional flow. The deflected jet flows down the wall as a semi-radial jet, and experiments show that u_{rm}/u_L varies between 0.3 and 0.7. Hestad [9] shows that u_{rm}/u_L is a function of jet width compared to the widt of the end wall. Large jet width corresponds to two-dimensional flow ($u_{rm}/u_L \sim 0.7$), and a small width of the primary jet corresponds to radial flow at the end wall ($u_{rm}/u_L \sim 0.3$).

The maximum velocity in the occupied zone u_{rm} can be obtained from equation (1) when the variation of u_{rm}/u_L is known

$$u_{rm} = u_o(\frac{u_{rm}}{u_L})K_a\frac{\sqrt{a_o}}{L+x_o} \qquad \text{(m/s)} \qquad (3)$$

Experiments in the subtask 1 work do not involve change in jet width or change in room width, and therefore it is not possible to add much new experience to the assumptions. The experiments show that u_{rm}/u_L is equal to 0.45 in the room geometry specified in subtask 1.

4. PENETRATION DEPTH OF NON-ISOTHERMAL JET

An undisturbed wall jet will penetrate the ventilated room in case of isothermal flow, and it will entrain air from the occupied zone to induce recirculating air movement in the room. This picture will change when a thermal load is supplied to the room. The supply temperature will be reduced and the load may reach a level such that the wall jet will separate from the ceiling at a distance x_s from the diffuser and flow down into the occupied zone as shown in figure 7.

Situations with a short penetration depth are undesirable because the jet may have a high velocity and a low temperature when it flows into the occupied zone, and a calculation of the penetration depth is thus a part of the design procedure of the air distribution system.

Grimitlin [11] has shown that the penetration depth for a cold three-dimensional wall jet is proportional to $1/\sqrt{Ar}$, where Ar is the Archimedes number. An analysis of the forces acting on a non-isothermal wall jet leads to the following equation

$$\frac{x_s}{\sqrt{a_o}} = 0.19K_{sa}K_a(\frac{1}{Ar})^{0.5} \qquad (4)$$

where K_{sa} is a constant dependent on parameters outside the wall jet, such as room dimensions, location of thermal load, etc.

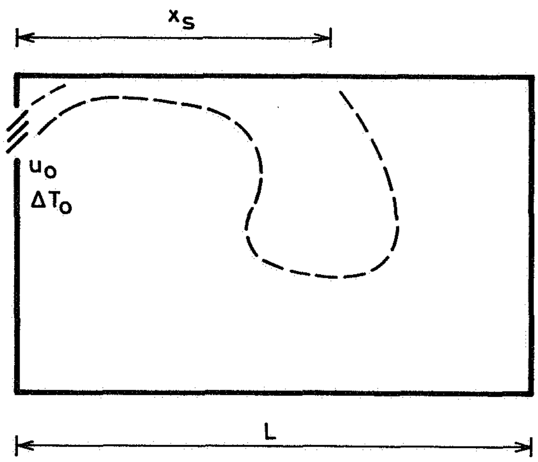

Figure 7. Penetration depth x_s of a thermal jet in a room.

Comparisons between measurements on different Air Terminal Devices and values calculated from equation (4) show a fairly reasonable agreement for $K_{sa} = 1.5$. The measurements are made in a large test area with floor mounted heat sources, see reference [12].

The influence of heat source location must be an important parameter. Some measurements in reference [9] suggest that K_{sa} should be equal to 1.6 for end wall mounted heat sources as used in the IEA test case E.

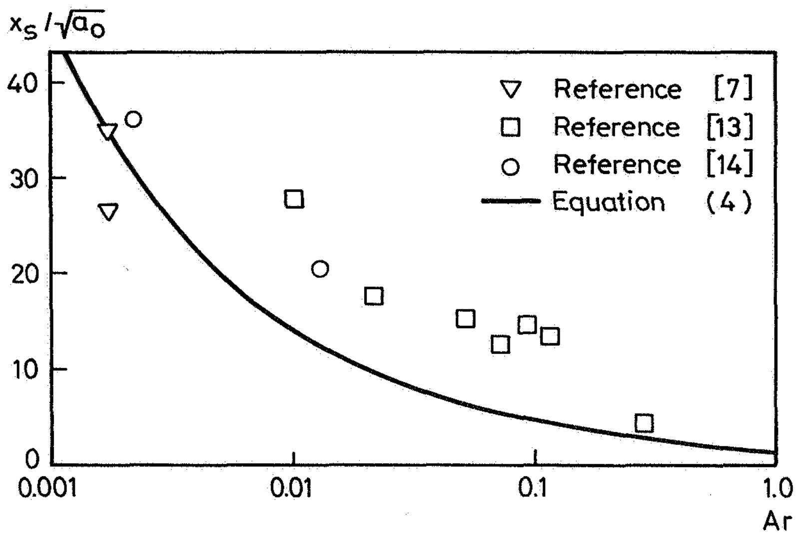

Figure 8. Measurements and predictions of penetration depth for the three-dimensional wall jet in the test case E. Room and diffuser are specified according to the requirements in references [3] and [4].

Figure 8 shows the measurements made by the following countries: Finland [7], Sweden [13] and Norway [14]. The measurements confirm the variation of the type $x_s \sim 1/\sqrt{Ar}$ but it is obvious that the penetration depth calculated according to equation (4) is underpredicted at high Archimedes' numbers.

Experiments in the subtask 1 work do not involve change in heat source location and room dimensions, so it is not possible to get insight into the shortcomings of equation (4). It can be concluded that equation (4) describes the general behaviour of a non-isothermal jet, but further research is needed to use this simplified model as part of a general design procedure.

Equations (3) and (4) can be used to make some general statements on the design of an air distribution system. The specific load in the room is expressed by

$$Q = \rho_o c_p \frac{a_o u_o \Delta T_o}{LW} \qquad (\mathrm{W/m^2}) \tag{5}$$

where ρ_o and c_p are density and specific heat, respectively. Equation (5) is rearranged by substituting u_o from the expression in equation (3) and ΔT_o from the expression in equation (4). x_s/L is restricted to a constant value (e.g. 0.5) due to comfort considerations. The specific heat load of the room will now have the following connection to the design variables

$$Q \sim \frac{u_{rm}^3}{K_a} \tag{6}$$

It is desirable to have an air distribution system which can handle a high heat load. Equation (6) shows that this is possible if a high maximum velocity in the occupied zone is tolerated, and also that the ability to handle heat load is proportional to the third power of this velocity. Therefore it is important that the design procedure is able to lay down a system which gives a velocity u_{rm} close to 0.15 m/s or close to another high design velocity.

The K_a-value is another important design parameter. Equation (6) shows that it is efficient to use an Air Terminal Devise with a low K_a-value. A low K_a value corresponds to a high initial diffusion and it is partly achieved by a semi-radial or radial flow in the wall jet below the ceiling. The Air Terminal Device in the subtask 1 work does have a high initial diffusion, and a semi-radial wall jet is generated in the ceiling regions, see references [2] and [4].

5. PREDICTION OF AIR DISTRIBUTION WITH A CFD-CODE

The aim of this chapter is to evaluate the CFD-method as a design tool in comparison with the simplified models. A general evaluation of CFD-mehods in connection with the subtask 1 work is given elsewhere, and the trend of the method is also given in reference [15].

Comparison with simplified models shows only a small gain if the design work is a question about obtaining the level of maximum velocity in the occupied zone and penetration depth of a non-isothermal jet in a room of conventional geometry.

Demands on the design work is increasing due to comfort and energy considerations. The CFD-method shows in this connection a number of possibilities compared to simplified models which will be discussed in the following

- the CFD-method gives a general description of the parameters in the room at design conditions as well as at other conditions,
- the CFD-method can predict some parameters which are important in the evaluation of the air distribution system but very time consuming to measure by full-scale experiments,
- the CFD-method may be the only practical design method in case of very large enclosures with complicated geometry and complicated boundary conditions.

Reference [5] shows an example on prediction of comfort level. It is expressed by the percentage of dissatisfied people because of turbulent and draught at foot level. The predictions show that half of the room has a draught risk about 16 - 20%, and the rest of the room has a draught risk which is below 16%. This type of information is general and useful compared to the statement that the maximum velocity in the occupied zone is 0.2 m/s. Furthermore, it is a type of information which reflects the accuracy of the method and an example of the general description of the situation which can be achieved by the CFD-method.

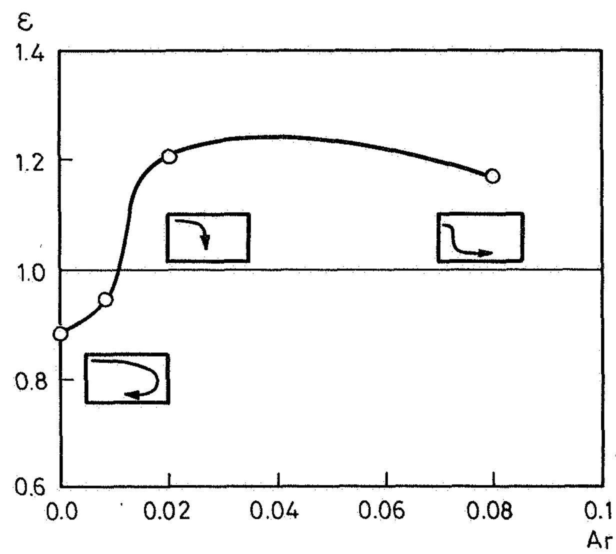

Figure 9. Ventilation efficieny versus Archimedes' number in a room with two-dimensional flow, see reference [15].

The prediction of ventilation efficiency is an example where the CFD-method is used to obtain results which would be very time consuming and expensive to obtain by full-scale experiments. Figure 9 shows the ventilation efficiency as a function of the Archimedes number in a room with a slot and two-dimensional flow. The predictions show that the ventilation efficiency is high in the case when the jet penetrates half of the room length, and this is also the flow pattern which is optimum for thermal comfort, taking into account that the heat load had to be removed from the room by the ventilation system. Computer predictions of this type are useful in the evaluation of an air distribution system and in the work on minimizing the energy consumption.

The most important area for the airflow simulation is undoubtedly the predictions of air movement in large enclosures with complicated geometry and complicated boundary conditions.

Large areas exclude the use of full-scale experiments. It is also difficult to use a simplified design method because the geometry may be complicated and there can be more sources for the air movement as for example diffusers, pressure difference around the building, cold downdraught and thermal plumes.

The glass covered "Hansa" shopping arcade in Turku is a typical example of a large enclosure with a complicated geometry, see figure 10. It is raised between a number of older buildings which include some warehouses, a hotel and the theatre of the town. The arcade may be looked upon as one large room and further research will show if it is possible to make any subdivisions of such a space. Figure 11 shows a part of the arcade or atrium between an old building and some of the new constructions. Is is obvious that the space also has a complicated geometry in the vertical direction.

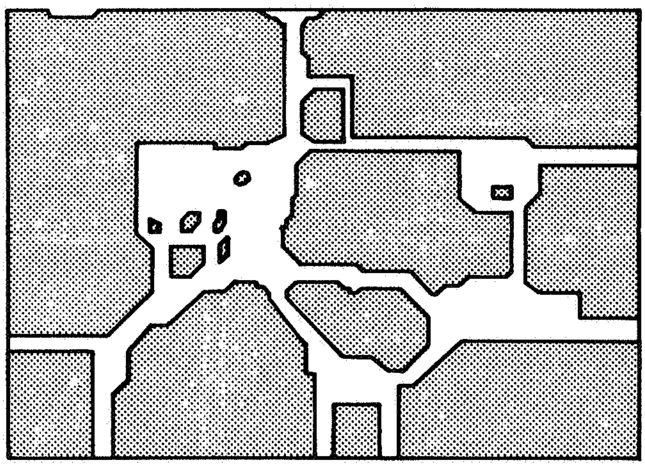

Figure 10. Horizontal section of the glass covered "Hansa" shopping arcade in Turku.

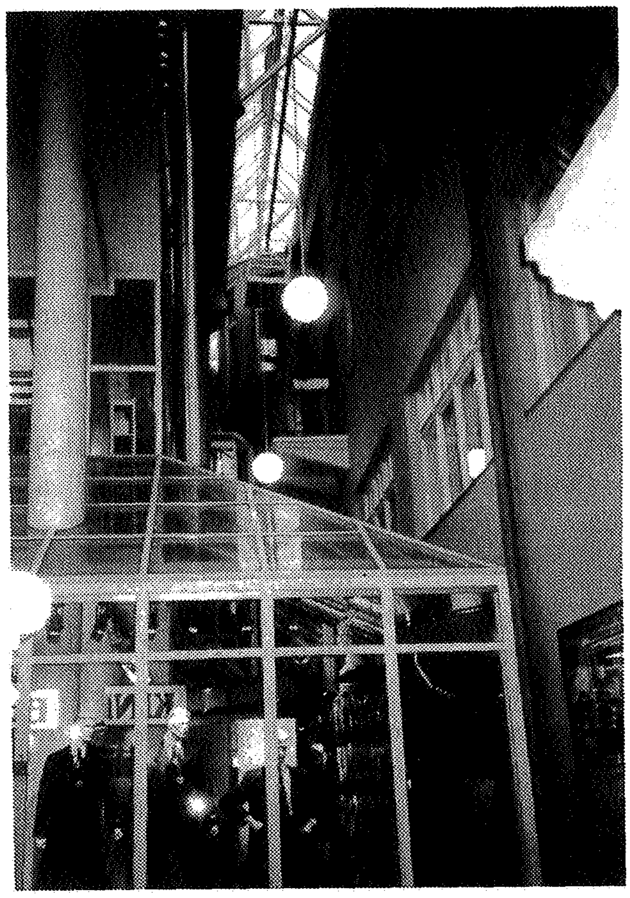

Figure 11. Details in "Hansa" shopping arcade.

The main purpose of the design work in large enclosures is to control the energy flow and temperature level. It is also very important to have a high ventilation efficiency in the occupied areas, and a system which can handle this area without too large air exchange in the rest of the air volume. Smoke movement in case of a fire and necessary escape route is an important subject. It is also necessary to limit the air velocity in the occupied areas because many people will work with restricted activity level in the shops and in open offices.

It is obvious that a CFD-method can be an important tool for the design of air movement in constructions as shown in figures 10 and 11. The CFD-method is able to handle many different sources as diffusers, pressure difference and temperature difference in a single calculation and is therefore able to predict the total effect of all sources.

In special situations when the space can be subdivided into conventional geometry and the flow can be described by parabolic equations it will also be possible to use simplified design models in large enclosures.

CONCLUSION

Simplified models are able to indicate design values as the maximum velocity in the occupied zone and penetration depth of a thermal jet in a room.

The design according to throw of an isothermal jet is a fully developed method which has a sufficient level of accuracy when it is used in a regular room. The producers of Air Terminal Devices can incorporate experience in the method by the selection of constants for the ATD and by the selection of throw and reference velocity.

A simplified model which is able to predict the real value of the maximum velocity in the occupied zone needs further development.

The prediction of penetration depth of a non-isothermal jet seems to be possible but is is necessary to do more experiments on different room geometry and heat source location.

Airflow simulation by a CFD-method is specially useful because the information is extended compared to the information obtained by a simplified model. Airflow simulation shows for example the distribution of velocities in the occupied zone as well as the design velocity. It is also possible to predict parameters which are difficult and time consuming to obtain by other models or by experiments.

The CFD-method is specially important in cases when the air distribution has to be predicted in large enclosures with complicated geometry and different sources for the air movements, as for example in shopping arcades, atria, factories, and theatres.

Generally speaking, simplified design models work well when the space can be subdivided into conventional geometry and the flow can be described by parabolic equations while a CFD-model can handle complex geometry and elliptic equations.

Measurements in different identical test rooms show significant deviation. This deviation seems also to correspond to the deviation between the simplified models and the CFD-method. In real installations, however, there will be larger deviations.

APPENDIX

The experiments in the subtask 1 work are made in test rooms with the dimensions $H \times L \times W$ equal to $2.5 \times 4.2 \times 3.6$ m. Only the Danish test room has a different height of 2.4 m.

The differences in height H will influence the measurements shown in figure 3, partly because they are made in sligthly different rooms and partly because the air exchange rate n is based on different room volumes. The practical importance is small because the difference in volume flow rate is within 4%.

The diffusers in all the test rooms are of the same type (HESCO type KS4W205K 370), and the individual difference between diffusers is small because it consists of injection moulded plastic parts. There may be differences due to the adjustment of the diffusers.

The diffuser area a_o is either measured as a geometrical area or it is calculated from flow rate and supply velocity, see reference [2]. The area has the following value

$$a_o = 0.0095\ \mathrm{m}^2, \text{ Geometrical area}$$

$$a_o = 0.00855\ \mathrm{m}^2, \text{ Effective area } (n = 3\mathrm{h}^{-1})$$

The effective area is used in figures 2 and 4 and the geometrical area is used in figure 8.

The Archimedes number is defined as

$$Ar = \frac{\beta g \sqrt{a_o}(T_R - T_0)}{u_o^2} \tag{A1}$$

except in figure 9, where the slot height is used as reference length. β and g are volume expansion coefficient and gravitational acceleration, respectively, and T_R - T_o is the temperature difference between return temperature and supply temperature. The velocity is given as $u_o = q_o/a_o$, where a_o is the geometrical area (figure 8).

REFERENCES

1. "Air distribution and air diffusion - Laboratory aerodynamic testing and rating of Air Terminal Devices"
Draft for international standard ISO/DIS 5219.

2. SKOVGAARD, M., HYLDGAARD, C.E., and NIELSEN P.V.
"High and Low Reynolds Number Measurements in a Room with an Impinging Isothermal Jet"
ROOMVENT '90, International Conference on Engineering Aero- and Thermodynamics of Ventilated rooms, Oslo, 1990.

3. LEMAIRE, A.D.
"Testrooms, Identical Testrooms"
Internal report for IEA Annex 20, TNO Institute of Applied Physics, Delft, 1989.

4. NIELSEN P.V.
"Selection of Air Terminal Device"
Internal report for IEA Annex 20, University of Aalborg, ISSN 0902-7513 R8838, 1988.

5. SKOVGAARD, M. and NIELSEN P.V.
"Modelling Complex Inlet Geometries in CFD - Applied to Air Flow in Ventilated Rooms"
12th AIVC Conference on Air Movement and Ventilation Control within Buildings, Ottawa, 1991.

6. BLOMQVIST, C.
"Measurements of Testcase B"
Internal report for IEA Annex 20, The National Swedish Institute for Building Research, Gävle, 1991.

7. HEIKKINEN, J.
Private communication, Technical Research Centre of Finland, Espoo, 1991.

8. EWERT, M. and ZELLER, M.
"Turbulence Parameters at Supply Opening (measurements)"
Internal report for IEA Annex 20, Rheinisch-Westfälischen Technischen Hochschule, Aachen, 1991.

9. HESTAD, T.
"A Design Method for Diffusers Based on Theory, Full-Scale Experiments and Practical Experience"
(In Norwegian), Tekniska Meddelanden nr. 83, Institutionen för Uppvärmnings- och Ventilationsteknik, KTH, Stockholm, 1975.

10. NIELSEN, P.V.
"Mathematical Models for Room Air Distribution"
Conference on "System Simulation in Buildings", Liege, 1982.

11. GRIMITLIN, M.
"Zuluftverteilung in Räumen"
Luft- und Kältetechnik, nr. 5, 1970.

12. NIELSEN, P.V. and MÖLLER, Å.T.A.
"Measurements on Buoyant Wall Jet Flows in Air-Conditioned Rooms"
ROOMVENT '87, International Conference on Air Distribution in Ventilated Spaces, Stockholm, 1987.

13. SANDBERG, M.
Private communication, The National Swedish Institute for Building Research, Gävle, 1991.

14. FOSSDAL, S.
"Measurement of Testcase E"
Internal report for IEA Annex 20, Norwegian Building Research Institute, Oslo, 1990.

15. NIELSEN, P.V.
"Airflow Simulation Techniques - Progress and Trends"
10th AIVC Conference, Espoo, 1989.