INTERNATIONAL ENERGY AGENCY
energy conservation in buildings and
community systems programme

Technical Note AIVC 33

A Review of Building Air Flow Simulation

March 1991

AIVC

Air Infiltration and Ventilation Centre
University of Warwick Science Park
Barclays Venture Centre
Sir William Lyons Road
Coventry CV4 7EZ
Great Britain

This **report** is part of the work of the IEA Energy Conservation in Buildings & Community **Systems** Programme

Publication prepared by
Annex V Air Infiltration and Ventilation Centre

Document AIC-TN-33-1991
ISBN *0* 946075 52 2

Participating countries:

Annex V
Belgium, Canada, Denmark. Germany, Finland, Italy, Netherlands, New Zealand, Norway. Sweden, Switzerland. United Kingdom, and United States of America

Distribution: Annex V only

Additional copies of this report may be obtained from:

The Air Infiltration and Ventilation ***Centre***
University of Warwick Science Park
Barclays Venture Centre
Sir William Lyons Road
Coventry CV4 7EZ
Great Britain

A Review of
Building Air Flow Simulation

Martin W Liddament

Contents **Page**

1 Introduction 1

2 Flow Representation **3**

2.1 Introduction **3**

2.2 **Laminar** Flow **3**

2.2.1 **Forced** Convection **3**

2.2.2 Mixed and Free Convection (Buoyancy) **5**

2.3 Turbulent Flow 5

2.4 Contaminant **Transport** 7

3 Computing the Flow Field 7

3.1 Introduction 7

3.2 Numerical Techniques **8**

3.2.1 Finite Volume/Difference Methods **8**

3.2.2 Finite Element Methods **9**

4 Discretisation **Systems** 11

4.1 Introduction 11

4.2 Uniform Orthogonal 12

4.3 Non Uniform Orthogonal **12**

4.4 Body Fitted Coordinates **13**

4.5 Local Mesh Refinement 13

4.6 Staggered Grid **14**

4.7 Grid Spacing **-** False Diffusion **14**

5 Boundary and Source Conditions **14**

5.1 Representation of Air Supply **15**

5.2 Representation of Infiltration and Natural Ventilation **16**

5.3 Representation of Heat **Sources** and Heat Loss 16

5.4 Pollutant Sources 17

5.5 Grid Representation of **Source Terms** 17

5.6 Flow **at Boundaries** 17

6 Validation 17

7 **Case Studies** 18

8 Conclusions 20

9 References **21**

Appendix 1 Equations and Nomenclature 33

Appendix 2 Research Algorithms *37*

Appendix 3 Commercial **Algorithms** 39

Preface

International Energy Agency

The International Energy Agency (IEA) **was** established in **1974** within the framework of the Organisation for Economic Co-operation and Development (OECD) to implement an International Energy Programme. **A** basic aim of the **IEA** is to foster co-operation among the twenty-one **IEA** Participating Countries to increase energy security through energy conservation, development of alternative energy sources and energy research development and demonstration (RD&D). This is achieved in part through a programme of collaborative RD&D consisting of forty-two Implementing Agreements, containing a total **of** over eighty separate energy RD&D projects. This publication forms one element of this programme.

Energy Conservation in Buildings and Community Systems

The IEA sponsors research and development in a number of areas related to energy. In one of these areas, energy conservation in buildings, the **IEA** is sponsoring various exercises to predict more accurately the energy use **of** buildings, including comparison of existing computer **programs,** building monitoring, cornparison of calculation methods, **as** well **as** air quality and studies of occupancy. Seventeen countries have elected to participate in this area and have designated contracting parties to the Implementing Agreement covering collaborative research in this area. The designation by governments of **a** number of private organisations, **as** well as universities and government laboratories, as contracting parties, has provided **a** broader range **of** expertise to tackle the projects in the different technology areas than would have been the case if participation was restricted to governments. The importance of associating industry with government sponsored energy research and development is recognized in the IEA, and every effort is made to encourage this trend.

The Executive Committee

Overall control **of** the programme is maintained by an Executive Committee, which not only monitors existing projects but identifies new areas where collaborative effort may be beneficial. The Executive Committee ensures that all projects fit into **a** pre-determined strategy, without unnecessary overlap or duplication but with effective liaison and communication. The Executive Committee has initiated the following projects to date (completed projects are identified by *):

I Load Energy Determination of Buildings*
II Ekistics and Advanced Community Energy Systems*
III Energy Conservation in Residential Buildings*
IV Glasgow Commercial Building Monitoring*

V **Air** Infiltration **and** Ventilation Centre
VI Energy Systems and Design of Communities*
VII Local Government Energy Planning*
VIII Inhabitant Behaviour with Regard to Ventilation*
IX Minimum Ventilation Rates*
X Building HVAC Systems Simulation*
XI Energy Auditing*
XII Windows and Fenestration*
XIII Energy Management in Hospitals*
XIV Condensation*
XV Energy Efficiency in Schools
XVI BEMS - 1: Energy Management Procedures
XVII BEMS - 2: Evaluation and Emulation Techniques
XVIII Demand Controlled Ventilating Systems
XIX **Low** Slope Roof Systems
XX **Air** Flow Patterns within Buildings
XXI Thermal Modelling
XXII Energy Efficient Communities
XXIII Multizone Air **Flow** Modelling

Annex V Air Infiltration and Ventilation Centre

The **IEA** Executive Committee (Building and Community Systems) has highlighted areas where the level of knowledge is unsatisfactory and there was unanimous aggreement that infiltration was the area about which least was known. ***An*** infiltration group was formed drawing experts from most progressive countries, their long term aim to encourage joint international research **and** increase the world pool **of** knowledge on infiltration and ventilation. Much valuable but sporadic and uncoordinated research **was** already taking place and after some initial groundwork the experts **group** recommended to their executive the formation **of** an *Air* Infiltration and Ventilation Centre. This recommendation was accepted and proposals **for** its establishment were invited internationally.

The aims **of** the Centre are the standardisation of techniques, the validation of models, the catalogue and transfer of information, and the encouragement of research. It is intended to be **a** review body for current world research, to ensure full dissemination of this research and based on a knowledge of work already done to give direction and firm basis for future research in the Participating Countries.

The Participants in this task are Belgium, Canada, Denmark, Germany, Finland, Italy, Netherlands, New Zealand, Norway, Sweden, Switzerland, United Kingdom and the United States **of** America.

1. INTRODUCTION

The pattern of air flow within a building or within individual zones or rooms can have a considerable impact on the energy performance of ventilation systems. In addition the behaviour of air flow influences the propagation of airborne pollutants, the thermal environment and general comfort conditions. In order to optimise design and to ensure a healthy interior, increasing attention is being focused on building air flow behaviour and mixing characteristics.

In many instances it is difficult to consider in detail the pattern of air flow or the influence of air movement on thermal and pollutant transport. Such factors as the interaction of leakage openings, the location of supply and extract terminals, room layout, occupant patterns and the location and strength of heat sources all combine to influence the pattern of flow (Figure 1.1).

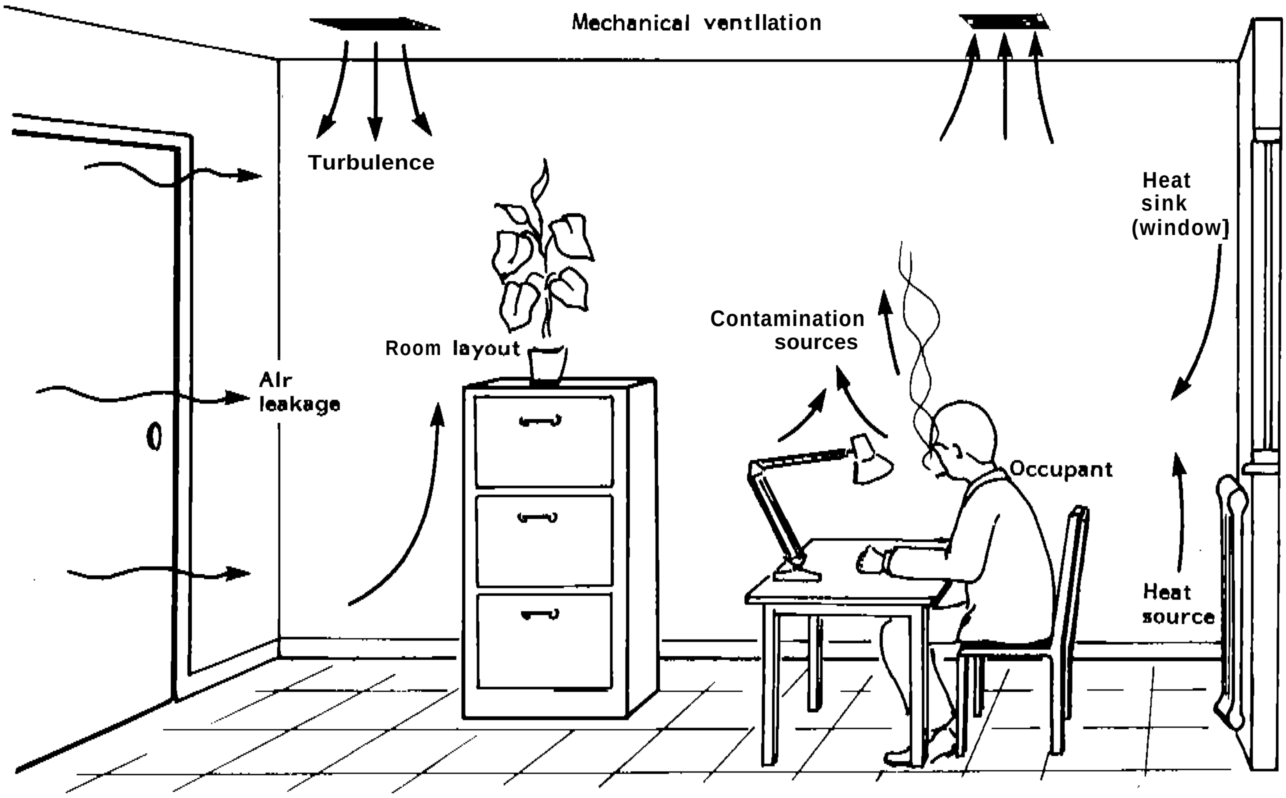
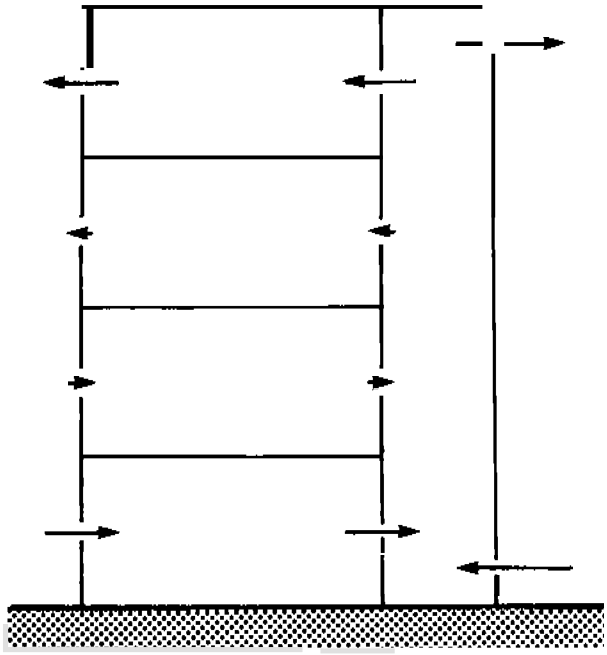


Figure 1.1 Factors influencing air movement

In the past, numerical techniques for building air flow analysis have tended to concentrate on the calculation of mass flow rates through defined openings and on the direction of flow into and out of zones, (Liddament, 1986). These methods provide information on interzonal air flow patterns and they may also be used to predict air exchange rates.

However, these methods ignore air flow behaviour within individual zones, since this is assumed to be instantaneously and perfectly mixed (Figure 1.2). Often, if more detail is required, ventilation effectiveness measurements may be made using tracer gas techniques. This approach is used to determine the degree to which fresh air is circulated and mixed within the occupied zone (Sandberg et al 1989), Sutcliffe (1990). Alternatively, anemometry measurements may be made, either within the building or within a suitably constructed **scale** model (Moog **1981,** Klobut **1985,** Ziemssen **1985,** Nielsen and Evensen et al **1987** and Heiselberg et al 1987, 1988).



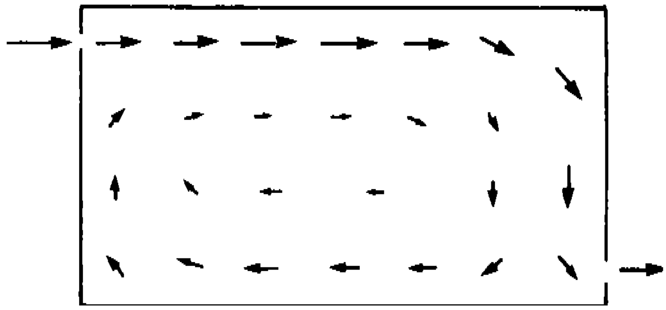


Figure 1.2

Interzonal Air Flow

Flow within each zone is undefined (conventional multi-zone model)

Figure 1.3

Room air flow

CFD modelling required (flow within zone defined)

Recently, increasing interest has been shown in the use of computational fluid dynamics to predict the pattern of air flow in buildings (Figure 1.3). Potentially, these techniques offer substantial benefits in the design and evaluation of both energy efficient and air quality efficient ventilation installations. However, as in many other areas of building physics, the successful implementation of these methods is substantially complicated by the individuality of building design and construction combined with almost limitless variations in building use, occupancy loading, and heating and ventilation requirements.

As a consequence, little is yet known about the validity and range of applicability of computational fluid dynamics code for building air flow analysis. These problems are currently being addressed by a number of IEA annexes and by other specialist groups in several countries.

The objective of this report is to outline recent developments in building air flow analysis and to focus on some of the difficulties associated with this complex field of study. Considerable developments in the area of computational fluid dynamics are currently taking place, especially in relation to refinements in calculation techniques. Rather than present an in-depth study of these developments, this report concentrates on the more general aspect of air flow in buildings. Bibliographic references are restricted to recent publications taken from the Air Infiltration and Ventilation Centre's Bibliographic Database - AIRBASE, which illustrate specific building examples and case studies. The appendices section contains references to related activities. These include a brief outline of the relevant flow equations, the results of the Centre's survey into the application of air flow codes for building air flow simulation, and summaries of selected public domain and commercial general purpose algorithms. An indication of the computer requirements for the use of these codes is also given.

2 FLOW REPRESENTATION

2.1 Introduction

Air movement occurs as a consequence of pressure differences between adjacent air masses which are sustained by naturally and/or mechanically derived forces. These forces may be time dependent, resulting in a variation of pressure distribution over time or they may be constant with respect to time, resulting in steady state conditions. In order to predict the pattern of air flow, the temperature field and the consequent distribution of pollutant within an enclosure, it is necessary to describe the equilibrium of forces at each point in the field which govern flow and transport behaviour. These expressions must take into account momentum, the convective and diffusive properties of air, friction and buoyancy forces, turbulence, the temperature field and the emission and distribution characteristics of pollutant sources. These concepts are introduced in this section, although no attempt has been made to present a full derivation. The intention is to illustrate the necessity of each parameter and to show how they combine to describe the total flow field. The basic concepts of flow analysis are introduced by considering laminar forced convection. Buoyancy production forces, turbulence and pollutant transport are then discussed.

2.2 Laminar Flow

2.2.1 Forced Convection

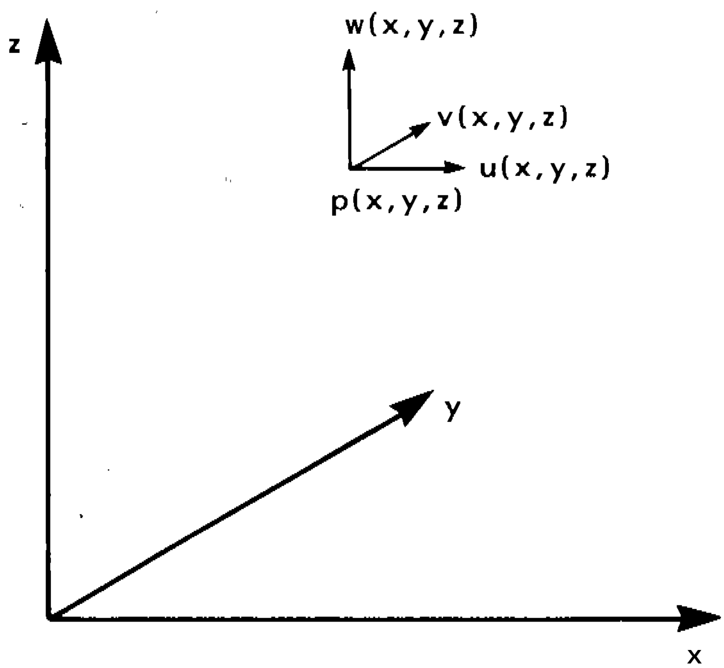
Laminar forced convection may be considered as undisturbed or smooth flow dominated by mechanical forces under isothermal (uniform temperature) conditions. Such flow may be represented by the Navier Stokes Equations. These express in mathematical terms the Law of Continuity and the Law of Conservation of Momentum for an incompressible fluid. Continuity simply states the conservation of mass, in which air entering a space is exactly matched by an equivalent mass of air leaving the space. The Law of Conservation of Momentum is a statement of Newton's Second Law of Motion as applied by considering the stresses associated with the motion of an element of an incompressible isotropic, Viscous fluid. The derivation of the Navier Stokes Equations is fundamental to flow theory and may be found in any standard reference text on this topic.

Assuming a steady state, three dimensional flow field, represented by a Cartesian Co-ordinate System in the x,y, and z directions, Figure 2.1, four unknowns are present at each location throughout the flow field; these are:

- Spatial velocity components in each of the three coordinate directions,ie u(x,y,z), v(x,y,z) and w(x,y,z).

- Spatial pressure distribution, ie p(x,y,z).

By expressing the Navier Stokes Equations for each of the coordinate directions and by applying the conservation of mass, a total of four equations are obtained and hence a solution to each of the unknowns is possible (Appendix 1). This represents the most basic set of equations for air flow analysis. Unfortunately it is not possible to solve these equations analytically and, instead, numerical methods must be applied. It is the development and application of such techniques that presents one of the principal challenges to flow simulation. A comprehensive introduction to these equations and an introduction to the principles of solution are presented by Shih (1984).



p = pressure; u, v and w are velocity values

Figure 2.1 Momentum and pressure fields acting at a point in a cartesian coordinate system

2.2.2 Mixed and Free Convection (Buoyancy)

If thermal gradients are present within **an** enclosure, conditions can no longer be regarded **as** isothermal; both the temperature and hence the density of air will vary throughout the enclosed volume. **For** practical purposes, density variations are small with respect to the mean density but are strongly influenced by the force of gravity. For this reason, if **a** small volume of air at a given temperature enters a region of higher temperature, it will descend under the force of gravity. This buoyancy force therefore only applies to the vertical component of motion and is represented numerically by including a gravitational term in the vertical component of the momentum equations. Nevertheless, it can have **a** considerable influence on the development **of** the flow field.

Two buoyancy induced convection conditions may be considered, namely mixed convection, in which buoyancy forces modify the pattern of forced air flow, and free convection, in which air flow is induced entirely by thermal effects. Numerically both conditions are treated identically.

By introducing buoyancy, two further parameters enter the flow field description, these being the temperature distribution, $T(x,y,z)$ and the density distribution, ρ (x,y,z). The temperature field is expressed in the form of **a** thermal transport equation based on the **Law** of Conservation of Energy, taking into account diffusion effects and thermal conductivity, while the density distribution is linked to the temperature and pressure fields by the Equation **of** State for an Ideal Gas.

A total of ***six*** unknowns or dependent variables are therefore now present in the flow field description, these being described by the following six equations:

- Continuity **-** Momentum (u component) **-** Momentum (v component) **-** Momentum **(w** component) **-** Thermal Transport - Equation of State for an Ideal **Gas**

2 3 Turbulent Flow

While the laminar flow formulation provides a relatively straightforward introduction to the basic principles of flow transport, it is limited to applications in which air velocity is low. Examples might include certain clean room ventilation and displacement ventilation systems. More generally, the behaviour of room ventilation is dominated by turbulence which is generated by relatively high supply air velocities and large temperature differences. Under these conditions, the smooth nature of laminar flow gives way to randomly fluctuating flow patterns superimposed on a mean velocity distribution. The concepts of free, forced and mixed convection are retained **as** for laminar flow. Essentially, the form of the Navier Stokes Equations remain the same but the velocity variable is replaced by time averaged **and** fluctuating components which yield additional stress terms (Reynolds Stresses). Similarly, additional turbulent temperature **and** pressure terms are introduced in the temperature and pressure equations.

To ensure a reliable prediction of the flow field under conditions of turbulence, it is essential that the scale of turbulence is accurately represented. This currently presents a particularly difficult problem, since the representation of turbulence relies substantially on empirically derived parameters which are difficult to quantify for room air flow applications. Several turbulence models are available and much research is currently taking place in this field.

A simple treatment of turbulence is based on a "zero" equation turbulence model in which no additional differential flow equation is required. Such an approach has been widely used in boundary flow analysis and has been applied to room air flow movement. This concept is based on the Prandtl Mixing Length Theory in which the magnitude of the turbulent Reynolds Stresses is equated with an eddy viscosity term analogous to the laminar viscosity associated with non turbulent flow. The advantage of such an approach is that no further equations are introduced to describe turbulence and hence the complexity of solution is reduced. Shih **(1984)** argues that, while this approach greatly simplifies the numerical analysis of turbulent flow, its validity for multi-dimensional turbulent flow no longer exists. However, Borgers and Akbari **(1984)** use a zero equation turbulence model based on the Prandtl Mixing Length Theory to study free convective turbulent transfer between parallel plates to simulate convective flow within a trombe wall channel. Various turbulent Prandtl Numbers and empirical constants taken from the literature were used. Based on limited experimental data it **was** estimated that 90% of the computed results were within 20% of the measured data.

More complex approaches are based on "2-Equation Turbulence Models". These models attempt to introduce additional transport equations of the same form **as** the Navier Stokes Equations which the Reynolds stresses must satisfy (Shih **1984).** Currently, the most commonly applied method is the ***K-E*** model in which the turbulent diffusivity is expressed in terms of the kinetic energy of turbulence, K, and the dissipation rate of kinetic energy of turbulence, ε (Launder and Spalding 1972). This results in two further transport equations which express the spatial distribution of K(x,y,z) and ε (x,y,z). These require simultaneous solution with the previous continuity, momentum and temperature components of the Navier Stokes and thermal transport equations. In addition the evaluation of **K** and ε require yet additional equations expressed in a similar form.

Originally this turbulence model was developed for high Reynolds Number or high turbulence flow. In room, however, low turbulence regions occur especially close to solid boundaries. Two approaches to dealing with low turbulence conditions tend to be used. Akbari et al **(1986)** give an example of the use of high Reynolds Number turbulence models combined with wall function corrections, since, it is argued,computing time is relatively speedy and storage needs are relatively low. This near wall or wall function correction is achieved by first defining **a** viscous sublayer above which flow is assumed to be fully turbulent. Within this sublayer, the mean velocity parallel to the wall or boundary is assumed to vary linearly with distance from the wall while,beyond this region, it is assumed to vary linearly with the logarithmic distance. Variations in turbulent kinetic energy, local turbulent shear stress and the dissipation rate of turbulent energy are then made consistent with these velocity functions. Within the viscous sublayer, turbulent shear stress is zero and the dissipation rate is constant. This approach also requires a near wall temperature model in order to reflect fluid temperature distribution within the viscous sublayer. Akbari applies a steady state energy equation for incompressible flow which is based on the assumption that the diffusion of heat in the direction of flow is small and that the mean velocity in the main flow direction is **a** function of only the distance from the wall.

Chen, Moser et al (1990) recommend the use of a "Low Reynolds Number **K**-ε model as developed by **Lam** and Bremhurst **(1981)** for predicting flow within rooms. The Wall Function approach of the previous technique separates the flow conditions of the fully turbulent regime and of the laminar sublayer by using different functions within each region. In the Low Reynolds Number approach, the ***K-E*** model **of** turbulence is retained but three functions, f1,f2 and fu, are introduced which are continuous and valid throughout the entire flow region. Within the turbulent regime, the value of each of these functions is unity, reflecting the fact that viscous forces within this region are negligible. However, **as** the laminar sublayer is approached, the value of these functions **vary** in order to reflect the increasing influence of viscous forces within this zone.

2.4 Contaminant Transport

A further requirement of air quality analysis is to include the distribution and propagation of pollutants from specified sources. This is approached by expressing pollutant flow **as** an additional transport equation analogous to the previous flow equations. Hence at least one more equation is needed to describe the pollutant field within the flow regime.

3 COMPUTING THE FLOW FIELD

3.1 Introduction

Clearly the simultaneous solution of the many equations needed to describe the fully turbulent, transient, convective flow field in the form of 3-dimensional partial differential flow equations, can be expected to require considerable computational effort. Large mainframe computers combined with lengthy processing times are commonly needed although much effort is currently being devoted to producing codes which will operate on smaller computers.

In order to **minimise** computational requirements, much can be done to simplify the number of equations and the number of dimensions included in the analysis. Thus, for example, the time dependent parameter is eliminated for steady state calculations. Similarly, it may be possible to reduce the problem from **a** 3-dimensional analysis to a 2-dimensional analysis. *Also* it is often unnecessary to solve the equations for all of the physical quantities. Under conditions of uniform temperature, for example, the thermal transport equation is not needed, while under conditions **of** laminar flow, the turbulence equations are eliminated.

Despite any possible simplifications, analytical solution of the transport equations is unlikely and ,therefore, numerical techniques must be applied. The purpose of this section is to outline some of these techniques.

3.2 Numerical Techniques

Numerical techniques used in air flow analysis include both finite difference and finite element methods. Whichever technique is applied, it is necessary to discretise the space into control volumes or elements and to use discretisation equations to represent the physics of flow within each of the elements (Section 4). In the case of finite difference or finite volume methods, the flow condition within each of the control volumes is represented by a single value. In the case of finite element techniques, the flow condition is represented by a functional relationship such that the value of the flow parameter is continuously variable throughout the discretised element. In both cases all parameters must be given initial arbitrary values from which the iteration process can commence. Iteration is a structured process in which the initial values of each flow parameter are adjusted until the flow equations balance. The process of reaching a successful balance point is known as convergence.

In order to solve the flow field, the individual flow or transport equations are first reduced to a generalised common form. This takes the form of a partial differential equation containing the following terms:

UNSTEADY + CONVECTION = DIFFUSION + SOURCE TERM TERM
TERM TERM

Hence flow is expressed in terms of an unsteady or time dependent component, a convective term, a diffusion term and a source term. By developing an algorithm to solve the general form of the flow equation, it follows that the same algorithm may be used to solve the specific transport equations.

3.2.1 Finite Volume/Difference Methods

The 'SIMPLE' Technique

Solving the flow equations is complicated by the fact that the various flow fields are interdependent and that none of them are initially defined. Further complications arise because the velocity components in each of the three coordinate directions are present in both the continuity and momentum equations. A method for overcoming these problems is described by Patankar (1980) using a finite difference/volume technique known as the Semi-Implicit Method for Pressure Linked Equations - SIMPLE. This method has subsequently formed the basis of many flow simulation models (see Appendix 2 and 3). The technique involves progressively improving upon an assumed pressure field, using a pressure correction formula, until the resultant component velocity values satisfy the continuity equation. An outline of the procedure is illustrated in Figure 3.1. This process is essentially a sequential approach. An initial pressure distribution is assumed and this is substituted into the momentum equations to evaluate an initial velocity distribution. A pressure correction term, p', is then applied and the consequent new pressure distribution and velocity distributions are evaluated. Other transport equations which influence the flow field (ie. temperature field, turbulence etc.) are similarly solved. The velocity values are inserted into the continuity equation which is then verified for compliance. This iteration process is continued until the continuity equation is satisfied.

Analysts describing recent applications of the "SIMPLE" method **or** derivatives of this approach include Awbi (1990) - ***ARIA,*** Chen et al (1987) and Kooi et al (1988) - CHAMPION, Chen et al 1990 - PHOENICS, Lemaire (1990) WISH3D. Many other examples are also covered in the references presented in this review. **In** addition Shih **(1984)** presents **a** demonstration **FORTRAN** Code based on the SIMPLE Technique while Whittle (1986) presents a comprehensive review of air flow simulation using this technique.

Multigrid Techniques

A problem with the sequential SIMPLE approach to solving the flow equations is the weak numerical coupling between the momentum and buoyancy forces. Changes in the temperature field during an iteration step are transferred to the momentum equation at the start of the following iteration. If the buoyancy "correction" during iteration is large, then it may not be possible to obtain a converged solution at all.

To overcome this problem, strongly coupled solutions are being introduced. Thompson et al (1987) and Vanka et al (1988) describe one such technique based on a "Multigrid approach". Discretisation follows that of the **SIMPLE** method in that a staggered grid is used in which velocities are associated with the faces of each control volume, while pressures and temperatures are associated with the centres of each control volume. However, instead of introducing a pressure correction equation, in order to initiate the sequential iteration process, the velocities and pressures are simultaneously updated, resulting in tight coupling between equations. However, since all the variables are held in computer memory at the same time, substantial memory is required. To further assist rapid convergence, **a** "multigrid" discretisation system is introduced in which part of the analysis is undertaken on a coarse grid and the results are applied to successively finer grids.

3.2.2 Finite Element Methods

Air flow analysis based on finite element methods has evolved from techniques used in structural analysis. Its particular advantage is in being able to apply an irregular grid which can be selectively refined and chosen to match the boundaries of the domain. Methods also tend to employ direct simultaneous solution **of** the continuity equation, the momentum equation and the remaining dependent variables rather than adopting a sequential approach. Such an approach demands substantial computer memory since information relating to all parameters need to be retained at the same time. However, the simultaneous approach ensures stronger coupling between the different flow parameters, with the result that convergence can be rapid. **A** further difficulty with the finite element approach is that, while an irregular grid system may be established within a 2-dimensional flow domain, it is difficult to establish such a finite element mesh within **3** dimensions. Thus progression from a 2 dimensional network to a three dimensional system is difficult.

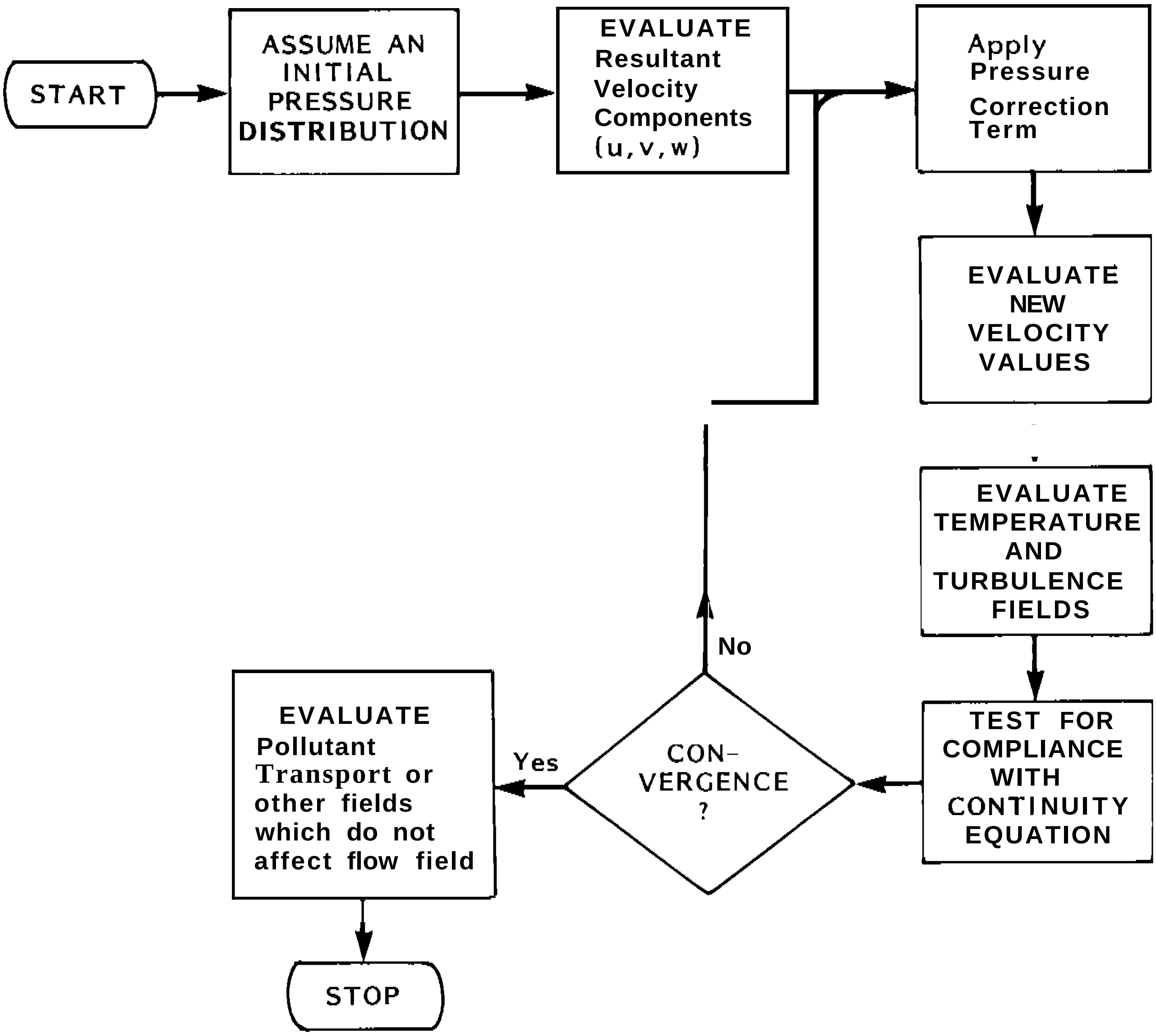


Figure 3.1 Outline of SIMPLE Method

Examples of the application of the Finite Element approach for the analysis of 3-dimensional turbulent air flow are described by Matsumoto et al (1985,1987) and Kerestecioglu **(1989).** This approach is also being developed by Baker(1990) for use **as** a potential ASHRAE Code.

Because of the way the discretisation equations have been formulated, it has generally been found that converged solutions from arbitrary starting conditions **is** more likely using finite difference techniques rather than finite element methods. In other words, when using finite element methods, more care is needed in establishing the starting conditions from which the iteration process is to proceed. However an advantage of the Finite Element method is that once appropriate starting conditions have been established, the process of convergence is much faster. Current developments in numerical analysis are focusing on combining the convergence reliability of the finite difference approach with the convergence speed of the finite element approach. An attempt to combine the advantages of the Finite Volume Approach with those **of** the Finite Element mesh is described by Lonsdale (1988) in his introduction to the **ASTEC** Code (Appendix **3).**

4 DISCRETISATION SYSTEMS

4.1 Introduction

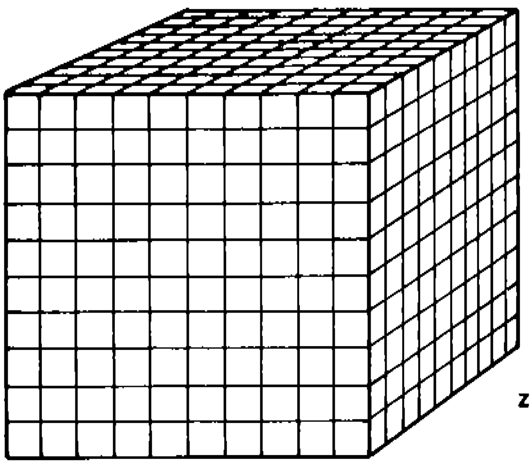
Discretisation is the method by which a space is subdivided into control volumes or elements to which the finite difference or finite element approximations **of** the transport equations may be applied. Careful consideration is needed with regard to the selection of an appropriate system. **A** technique is required which identifies each element within the discretised space and identifies its relationship with adjacent cells. *Also,* attention is required with respect to the coarseness of the network, the representation of boundaries and coping with regions within the network in which high velocity, temperature or pollutant concentration gradients occur. Other considerations include the availability of memory within the computer and the processing capability of the computer. **A** range of grid systems **has** been designed to cope with these various aspects; these include:

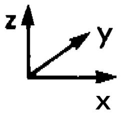
- Uniform Orthogonal Grid
- Non-Uniform Orthogonal Grid
- Body Fitted/Curvilinear Coordinate System
- Local Mesh Refinement

In conjunction with these, **a** "staggered grid" system is often applied in which **a** network "displaced" from the main grid is used in the calculation of the velocity field. Each of these systems **is** described below and is illustrated in Figures 4.1 to 4.5 respectively.

4.2 Uniform Orthogonal

A Uniform Orthogonal Network is established by subdividing the space or volume of interest with a series of equally spaced orthogonal lines along each coordinate axis (Figure **4.1).** This represents the most straightforward method of discretisation and results in a network of uniformly sized cube shaped cells or control volumes. The principal advantage of this technique is the relative ease with which the flow equations may be discretised and solved. Its main disadvantage is that it does not provide an opportunity to concentrate cells at locations where special detail is required, hence a detailed analysis involves introducing a large number of cells throughout the entire space. This results in **a** substantial increase in computer processing and memory requirements and may restrict the size of the problem which can be analysed. Also representation of the boundaries or of detail within the enclosure may prove to be difficult or inaccurate.





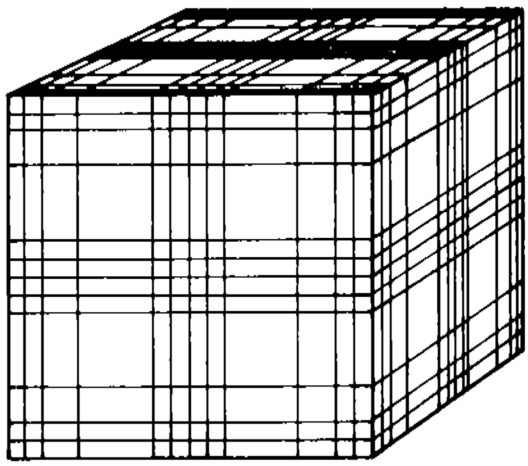


Figure 4.1 Uniform orthogonal grid **Figure 4.2 Non-uniform orthogonal grid**

4 3 Non Uniform Orthogonal

This represents an attempt to concentrate cells in regions where special attention is needed and, at the same time, to reduce the number of cells in regions of relative unimportance. Orthogonal lines in each of the three Cartesian coordinate directions are again used to divide the space but these are no longer uniformly spaced (Figure **4.2).** This approach is **commonly** used, although additional algorithms are required to ensure that the volumes and surface areas of each face of each of the cells are correctly identified. Boundary or interior detail may still be difficult to represent.

4.4 Body Fitted Coordinates

This is a method by which the grid system may be distorted such that it is no longer orthogonal (Figure 4.3). By distorting the network, boundaries and other features may be much more accurately represented than is possible when using an orthogonal grid system. The application of this approach in CFD codes is described in detail by Bum et al (1986). A similar technique , defined as generalised curvilinear coordinates is described by Murakami, Kato et al (1989). In each case, transformation equations are used to relate the Cartesian velocity components at the centre of mass of each control volume with the velocity components normal to the faces of the mass control volume. Significantly greater amounts of memory are required than is necessary for orthogonal systems to deal with the distorted grid and associated transformation equations.

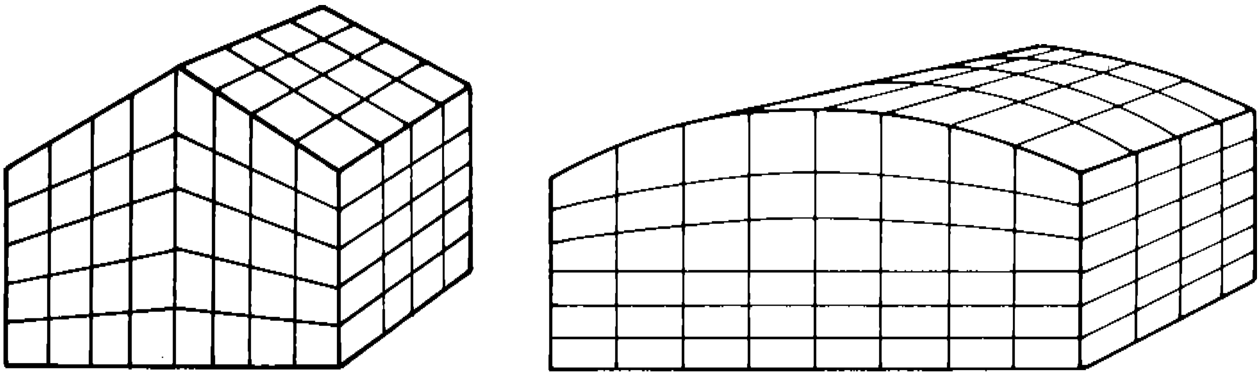


Figure 4.3 Body fitted/Curvilinear grid

4.5 Local Mesh Refinement

An alternative method to improve resolution at locations of interest is to introduce local mesh refinement (Figure 4.4) in which a series of control volumes are subdivided into smaller units. This results in an overhead on computing which is needed to identify refined regions and to handle the flow conditions from the coarse network to the refined region, however, it enables specific regions to be examined in detail.

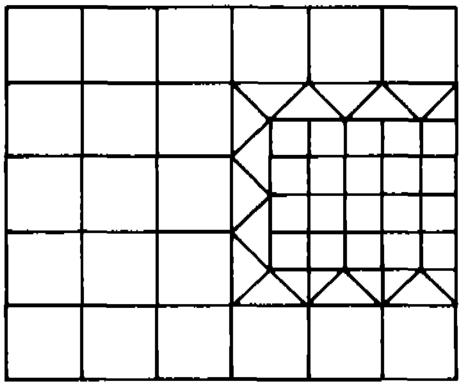


Figure 4.4 Local mesh refinement grid

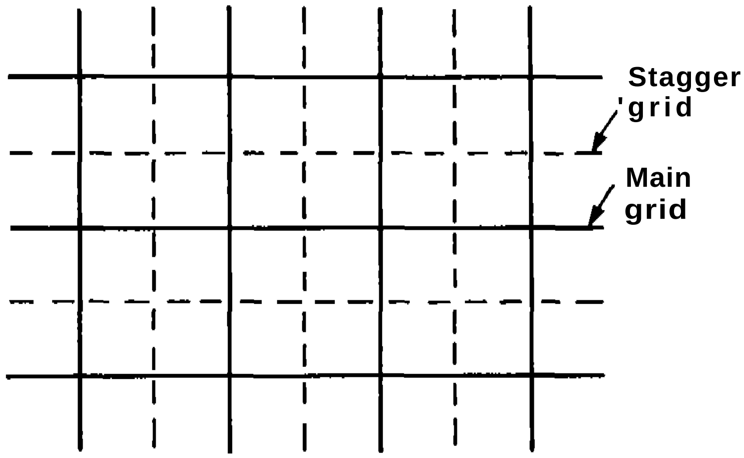


Figure 4.5 Staggered grid

4.6 Staggered Grid

Discretisation of the Transport Equations results in terms being expressed at alternate grid points rather than at adjacent points. This can result in poor accuracy and unrealistic flow representation (Patankar 1980). This problem can be overcome by using a staggered grid (Figure 4.5) in which the velocity values are calculated on **a** grid which is displaced from the main grid. In the Figure, the dashed line represents the boundaries of the "control volumes" around each of the main grid points. **It** is on these faces that the velocity components are calculated.

The staggered grid system is used in conjunction with **all** the previously described discretisation systems and is a principal feature **of** the SIMPLE method of solution. The use of non-staggered grid systems is described by Burns et al (1986).

4.7 Grid Spacing - False Diffusion

The selection of mesh spacing is very important. **A** close mesh substantially increases the number of control volumes and hence processing **and** storage needs. On the other hand, **a** coarse grid may lead to grid dependent and hence inaccurate results. Essentially, the Continuity Equation is satisfied but the flow field is incorrect. **A** reason for this is "false diffusion", in which the network and/or finite difference approximation is unable to resolve accurately the flow field. Instead ,a sharp transition at a location in the flow field, for example, is "smoothed out" and the resultant error is propagated throughout the rest of the flow field. **A** perspective on false diffusion is presented by Patankar (1980).

5 Boundary and Source Conditions

The pattern of flow within an enclosure is driven by conditions at the boundary and by the characteristics of source terms within the enclosed volume. In order to establish the correct flow pattern, it is necessary to represent **as** accurately as possible such terms within the numerical flow network. **It** is also necessary to ensure that flow adjacent to boundaries follows **boundary** layer theory. **A** recent review outlining boundary conditions **and** other aspects **of** Computational Fluid **Dynamics** is presented by Awbi(**1989).**

5.1 Representation of Air Supply

Air supply must be represented in terms of inlet **flow** velocity, volume flow rate and turbulence characteristics. These parameters may be obtained by test or from manufacturers data, although there is currently considerable discussion on the validity of such data, Standard testing includes the recommendations of British Standard **4773** for the measurement **of** throw and flow rates **of diffusers.** These tests, however, **relate** to flow in infinite enclosures. For this reason, manufacturers **also** provide typical **data** for various room configurations. In restricted environments, the flow pattern may be dominated by the characteristics of the enclosure rather than the diffuser itself. Examples **of** the influence of obstacles on the performance of jets are presented by **Awbi** et al (1987).

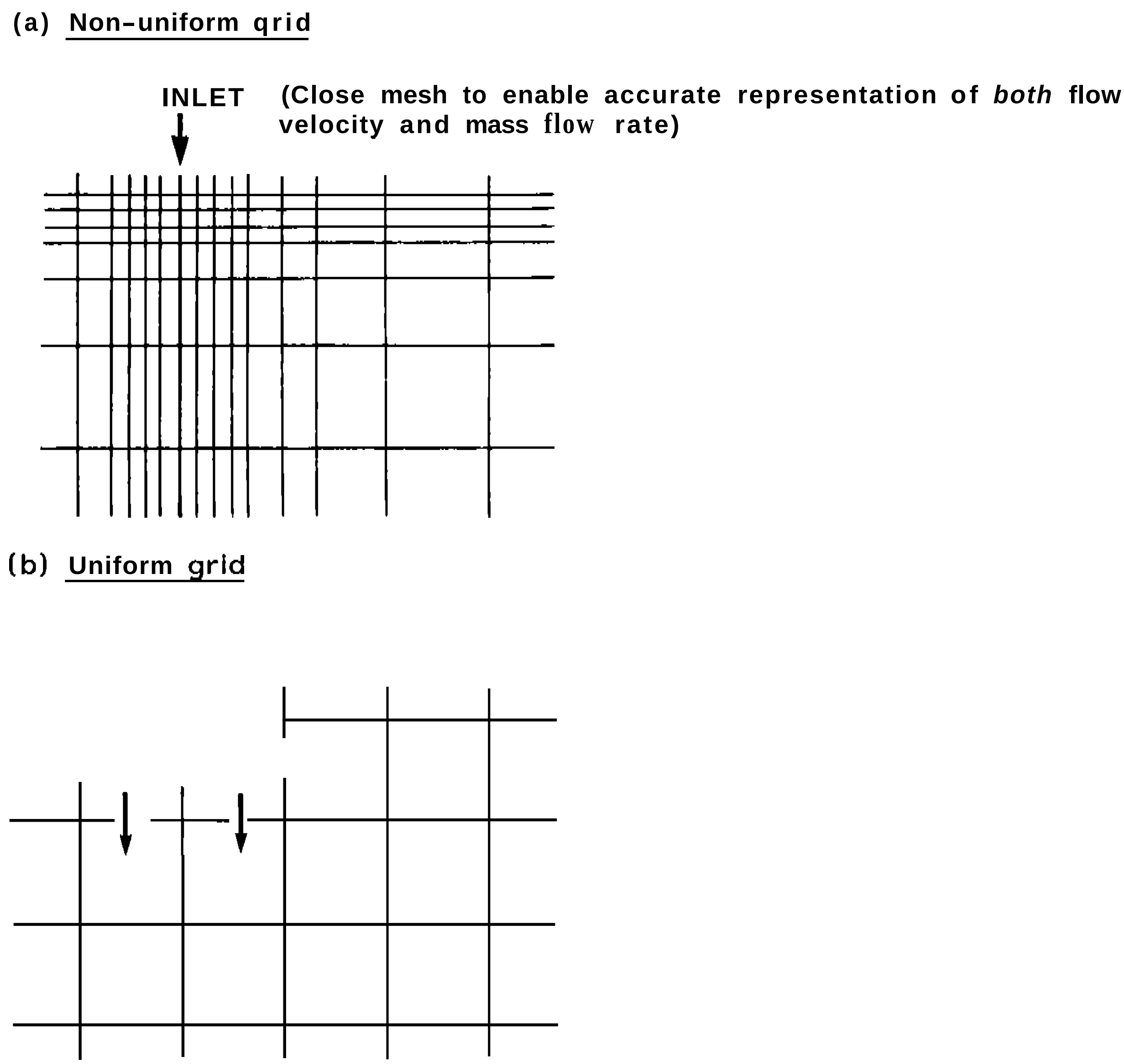


Figure 5.1 Representation of air supply

If the momentum field is to be accurately simulated, it is essential that both the velocity and volume flow rate are simultaneously represented. This requirement can impose severe demands on the grid system, since it dictates the coarseness of the grid at supply locations. An inlet, for example, supplying air at a velocity of 15 m/s at a volume flow rate of 0.15 m^3/s needs to be represented by a boundary control volume with a grid spacing of 0.15/15 m ie. .01m. Extended throughout an entire enclosure, such small grid sizes could result in an excessive number of grid points for even the most modest sized enclosures. Several authors have concentrated on this particular problem. Reinartz and Renz (1984) investigate the behaviour of a jet emerging from a radial air distributor using a 50x50 2-dimensional flow network and a solution technique based on the **SIMPLE** method. This network was regarded as sufficient to cover both the jet region and the flow field within the room. Alternatives include introducing a non uniform grid system, in which supply locations are represented by a much finer grid (Figure 5.1a), or representing supply flow at some distance from the opening itself at a location where flow rates are substantially reduced (Figure 5.1b). This latter approach offers considerable advantages and is discussed by Nielsen (1987).

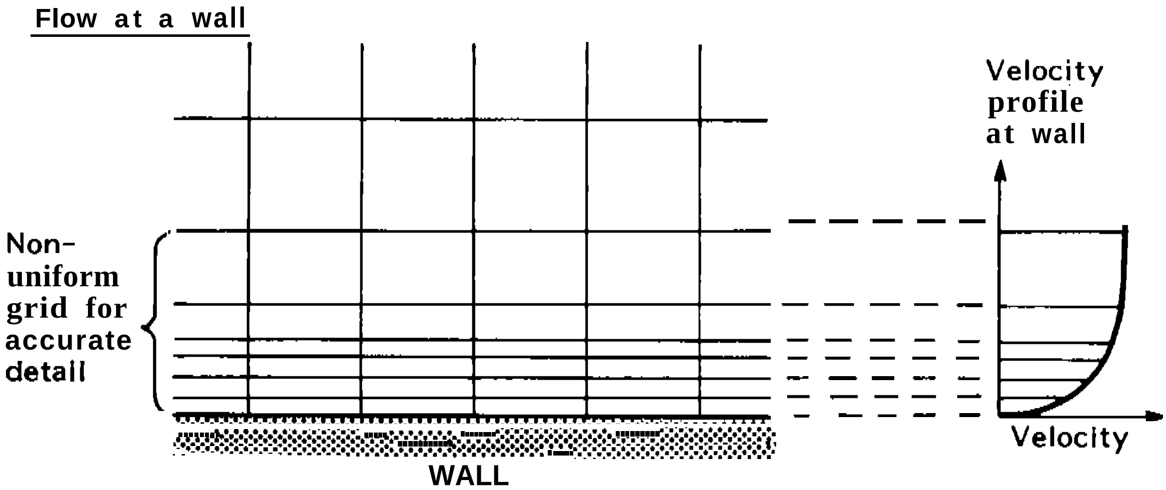


Figure 5.2 Boundary conditions

5.2 Representation of Infiltration and Natural Ventilation

If air flow is dominated by infiltration or natural ventilation, then the infiltrating flow rate must be included as part of the boundary source terms. It is assumed that such inflow will be widely distributed and at a low velocity, and hence may be represented by a coarse grid.

5 3 Representation of Heat Sources and Heat Loss

In the same way that the reliable simulation of momentum generated flow is dependent on the accurate representation of source flow characteristics, the simulation of buoyancy generated flow and the transfer of heat between boundaries is dependent on the accurate representation of the thermal properties of boundaries and heat sources. In the case of convective sources, this is achieved by expressing the thermal characteristics of sources in terms of temperature and heat transfer coefficients or U-values. Examples of the inclusion of convective sources within numerical models are described by Kurabuchi et al

(1990). Radiant energy sources can present particular problems, especially in high temperature flow analysis where direct transfer of radiant energy to the air stream can take place according to the absorption characteristics of the air. Such a transfer is normally insignificant in room air flow analysis and radiative exchange is normally assumed to take place directly between the source and adjacent surfaces as described by classical black body radiative exchange. The subsequent transfer of energy to buoyancy generated flow is through convective exchange between the resultant heated surfaces and the surrounding air stream. An approach to the simulation of radiative heat exchange is described in detail by Shih (1984).

5.4 Pollutant Sources

Pollutant sources are described by means of emission characteristics and location within the grid. Source terms may be constant or time dependent.

5.5 Grid Representation of Source Terms

Within the enclosure itself, source terms such as heat supply and pollutant generation are included by identifying the relevant grid location and specifying the source value. As the iteration proceeds, values at source locations are maintained at their predefined static or time varying settings.

5.6 Flow at Boundaries

Flow adjacent to boundaries must obey boundary layer theory, thus there is no tangential velocity at the boundary surface. Within the boundary layer which develops outward from the surface, velocity is influenced by shear stress and surface drag. Under conditions of laminar flow, the shear stress is given by the laminar velocity multiplied by the velocity gradient. Under turbulent conditions, the velocity profile follows a Log Law relationship instead of a linear relationship. The type of flow at the boundary proximity is determined on the basis of a local Reynolds Number. These problems are further described in Section 2.3. To cope with the steep change in velocity at boundaries a non uniform grid may be needed (Figure 5.2).

6. VALIDATION

A number of examples of the validation or evaluation of air flow simulation code may be found in the literature. It is also an important objective of IEA Annex 20 (Moser 1989). Measurement methods are, in addition, being used in conjunction with numerical methods as a technique to fine tune or adjust model parameters. Kooi and Chen (1986) give examples of the use of measurement results for the fine tuning of various wall functions applied to PHOENICS and CHAMPION code. Measurements for validation purposes tend to be difficult to control in full size buildings and hence much is being performed on scale models or test chambers. Smoke is widely used as a qualitative measure of air flow and has been used as a comparison against solutions to the Navier Stokes Equations (Ziemssen 1985). For quantitative results, however, air flow within test chambers is measured using anemometry. Examples include Murakami et al (1987,1988,1989,1990), in which turbulence modelling results were compared with measurements made in a 1/6th test chamber for various configurations of ventilation internal room obstructions and contaminant sources. In each case a tandem type, parallel hot-wire anemometer was used to determine the vector components of turbulent flow.

Also visualisation of flow across any section was made possible by using **a** laser light sheet combined with magnesium carbonate powder. Good correspondence between calculation and measurement was observed.

Chen has produced a series of papers comparing results with measurements for **a wide** range of test chamber configurations. In comparative studies involving the measurement of air flows at low velocities in rooms, Chen **(1989)** points out that mean velocities arising from mixed convection can be small in relation to the turbulent intensity and that hot wire anemometry could only be used at locations where velocity **was** relatively high. Velocities below 0.05 m/s were found to be difficult to measure because of calibration and heat transfer problems. Despite measurement difficulties, generally good comparisons between measurement and calculation have been achieved. In an another study, Chen, Kooi and Meyers (1988) report good agreement between calculations using the **PHOENICS** Code and measurements of air flow and temperature distribution in a full scale climatic chamber using different supply systems.

Pericleaus et al (1988) present results in which JASMINE and **PHOENICS** code **(Cox** 1986, Appendix **3)** was used to simulate the transient behaviour of smoke and fire spread in a 1/6th scale model of a semi rigid sports hall. Model dimensions were 11.6m high, **34m** long and 28m wide, therefore this represented a substantial sized building for air movement tests. Measurements included both smoke movement and temperature fields. Quantitative agreement was found to be good except in the immediate vicinity of the fire source. Inaccuracy within this region was associated with the representation of the source by **a** relatively coarse grid.

Awbi and Nemri (1989) compare numerical prediction with experimental data for a two dimensional isothermal and non isothermal flow regime and use the results **as** a guide for more complex simulation regimes. Comparisons between calculation and measurement were based on the use of **a** test chamber of 4.2m square section and a height of 2.8m. Calculations were found to correlate well with measurement away from the boundaries but discrepancies close to the boundary of the chamber were associated with the inability of the ***K-E*** model to represent **flow** at low Reynolds Number. Under non isothermal conditions, agreement between predicted and measured flow and temperature profiles was found to be close **for** most practical purposes.

7. CASE STUDIES

The application of Computational Flow Dynamics to building **air** flow problems **as** part of the design process is relatively new, although some examples are referenced in the literature. **A** few such examples are illustrated in this section to give an indication of the scope of applications of CFD Techniques.

Broyd et al(**1983) give** examples of the **use of a** 2-dimensional CFD **code** (CAFE) in the assessment of safety and air quality in industrial buildings. Four examples are cited covering coal storage, a chemical plant, a clean room **and** the turbine hall of **a** nuclear power station. **A** further example **of** clean room design using this code is **also** described by Broyd, Deaves **and** Oldfield **(1983)** in which computational results are supported by full scale measurements. Markatos (1983) gives an example of the use of 3-dimensional PHOENICS (Appendix **2)** in the analysis of air flow and heat transport in **a** television studio. **A** purpose of the study was to determine an optimum ventilation configuration for acceptable flow velocities and temperatures within the occupied zone of the studio.

In a series of papers Chen and Kooi (1988), Chen (1989) and Chen, Meyers and Kooi **(1989)** describe the analysis **of** 3-dimensional flow in an office type room with cooling using **CFD** analysis combined with experimental verification. Energy analysis is conducted using a PC computer program called **ACCURACY.** The purpose of this investigation was to evaluate the performance of various heating and ventilation systems in relation to both energy requirements and comfort conditions. In further studies by Kooi and Chen (1989) and Chen, Hoonstra and Kooi (1990), the performance of displacement ventilation systems are evaluated in relation to air quality and energy use. It is concluded that for cooling systems **a** displacement approach could require 26% less energy than an equivalent mixing ventilation system.

Other studies involving the evaluation of ventilation systems include Murakami, Kato and Suyama (1989), who report on an investigation into the influence on air **flow** of supply and exhaust arrangements. Particular emphasis was placed in clean room design. While the arrangement of exhaust points was found to have little effect on the flow field, such changes were often predicted to have **a** large influence on the contaminant diffusion field.

Ishizu **(1986)** also makes an evaluation of ventilation system using CFD techniques. He studied the optimum positioning of air inlets in relation to outlet location in order to maximise the efficiency of ventilation.

A number of case studies have been concerned with the application of computational fluid dynamics in smoke movement analysis. Pericleous et al (1988) give details of the simulation and comparison **of** smoke movement in **a** sports stadium **as** part of an investigation into predicting fire and smoke spread. In another example Waters (1989) describes the use **of** flow simulation techniques for fire and smoke movement and applies these techniques to the Stansted Airport Terminal Building in the United Kingdom and to the Lloyds Building in London, **UK.**

Free convection and heat transfer has also been the subject **of** detailed analysis using air flow simulation methods. Chen, Ho and Humphrey (1987) describe the simulation **of** convective flow in an enclosure at high levels of buoyancy. Borgers and Akbari (1984) and Akbari et al (1986) investigate modelling methods for air movement and heat transfer in passive solar design.

In a recent development Haghitat et al (1989) have reported on a numerical study investigating natural convection and air flow patterns within a partitioned enclosure subjected to turbulent flow. Thus an attempt has been made to combine multi zone air flow with air movement within individual zones. Numerical results indicate that the flow patterns are substantially influenced by the size and location of the interzone opening.

Huber et al (1990) illustrate the use of CFD PHOENICS code in the development of CEN Standard 130 on a proposed European test chamber for radiators. It is shown that it is possible to calculate the air flow pattern around a radiator with **a** known convective heat flux but computing time is substantial.

8. CONCLUSIONS

Computational Fluid Dynamics provide an opportunity to evaluate air flow patterns in buildings. In addition to air flow analysis, such techniques may be used to predict the propagation of pollutants, fire and smoke, and to evaluate flow velocities and temperature distributions. These methods therefore have potentially important applications in the design of the interior environment. However, although such methods have been widely applied in other areas of engineering flow analysis, their application to building physics is relatively new and has been restricted to only a few studies. Additionally, these codes have yet to be fully verified for use in this area of analysis. Especially, much further research is needed in relation to representing turbulence in rooms and in representing boundary conditions, obstacles and diffusers.

The complex nature of flow analysis is such that these numerical methods can be expensive to apply. Substantial computer hardware and processing time may be needed to analyse complex flow problems. A high degree of specialist knowledge is also required. For these reasons suppliers of software also normally provide a consulting service. CFD software should be regarded as a tool which will assist in design and analysis but will not compensate for lack of expertise. Future software developments could usefully focus on providing the necessary knowledge base and user interface needed to provide general accessibilty.

Many advances are currently taking place in the development of solution techniques and discretisation systems. In some instances these developments make further demands on hardware requirements. The relative benefits of these developments with regard to building air flow analysis need to be assessed as part of a validation study. Very little evidence exists on the "blind testing" of codes. A sensitivity analysis of parameters is also needed so that the key parameters may be identified and incorporated into simplified methods. Many of these items are being addressed by IEA Annex 20 "Air Flow within Buildings" (Moser 1989) and by ASHRAE Research Project 464 "Calculation of Room Air Motion" (Baker 1990). Future applications for air flow prediction techniques may be expected to include perceived indoor air quality, expressed in terms of odour, assessments of draught risk and discomfort due to draughts, and indicators of local air quality such as air change efficiency and ventilation effectiveness. In turn, these studies can be expected to reveal the optimum ventilation strategies necessary to satisfy fresh air requirements in the most cost effective and energy effective way.

9 References

#NO 3838 Development of a turbulent nearwall temperature model and its application to channel flow.
AUTHOR Akbari H, Mertol A, Gadgil **A,** Kammerud R, Bauman F
BIBINF Fed Rep Germany, Warmeund Stoffubertragung, No.20, 1986, **pp189201.** #DATE 00:00:1986 **in** English
ABSTRACT A numerical study of fluid flow and heat transfer in a twodimensional channel under fully developed turbulent conditions is reported. A computer program which is capable of treating both forced and natural convection problem under turbulent conditions has been developed. The code uses the highReynoldnumber form of the two equation turbulent model (ke) in which a turbulent kinetic energy nearwall model is incorporated in order to accurately represent the behaviour of the **flow near** the wall, particularly in the viscous sublayer where the turbulent Reynolds number is small. A nearwall temperature model has been developed and incorporated into the energy equation to d o w accurate prediction of the temperature distribution near the wall and, therefore, accurate calculation of heat transfer coefficients. The sensitivity of the prediction of flow and heat transfer to variations in the coefficients used in the turbulence model is investigated. The predictions of the model are compared to available experimental and theoretical results; good agreement is obtained. The inclusion of the nearwall temperature model has further improved the predictions of the temperature profile and heat transfer coefficient. The results indicate that the turbulent kinetic energy Prandtl number should be **a** function of Reynolds number.
KEYWORDS mathematical modelling, air flow

#NO 3856 Air jet Interference due to ceilingmounted obstacles.
AUTHOR Awbi **H** B, Setrak A A
BIBINF Sweden, Stockholm, Proceedings of Roomvent 87,1012 June 1987, 15pp, 11 **figs,** 20 refs. #DATE 00:06:1987 in English
ABSTRACT The effect of surfacemounted single and multiple obstacles in the path of a twodimensional wall jet is investigated experimentally and theoretically using a finite difference solution of the flow conservation equations. It is found that both the height, measured from **the** surface, and depth along the flow of an obstacle affect the distance from the supply slot at which the jet is about to separate, ie the critical distance. The presence of an obstacle accelerates the decay of the jet and the decay increases further when the jet separates from the surface. With multiple obstacles the diffusion of the jet increases **as** the relative height of the obstacles d/h increases.
KEYWORDS air flow, **ceiling**

#NO 3333 Application of computational fluid dynamics in room ventilation.
AUTHOR Awbi **H** B
BIBINF Building and Environment, Vol24, No 1,1989, pp7384, 15 **fig,** 34 refs. #DATE 00:00:1989 in English
ABSTRACT **This** paper presents the results of a computer program developed far solving 2 and 3D ventilation problems. **The** program solves, in finite difference form, the steadystate conservation equations of mass, momentum and thermal energy. Presentation of the fluctuating velocity components is made **using** the ke turbulence model. Predicted results of air velocity and temperature distribution in a room are corroborated by experimental measurements. The numerical solution is extended to other room ventilation problems of practical interest.
KEYWORDS cornputer, mathematical modelling, rooms, ventilation

#NO 3867 Scale effect in room air movement modelling.
AUTHOR Awbi **H** B, Nemri M M
BIBINF Yugoslavia, Proceedings **CLIMA 2000,** Aug 27 Sept 1, 1989, 6pp, 3 figs, 10 refs. #DATE 00:09:1989 in English
ABSTRACT In this paper, a numerical procedure is applied for solving the twodimensional NavierStokes equations describing the flow in **an air** conditioned room using the finite volume method. The effect of turbulence is described by the ke turbulence model. The range of influence of Archimedes and Reynolds numbers on the **air** velocity and temperature distribution in the room is investigated **using** the numerical solution. Comparison of the numerical prediction is made with experimental data. The results of the numerical solutions **can** be used **as** a guide for the physical modclling of **air** movement under more complex thermal conditions.
KEYWORDS air flow, modelling

#NO 4126 The role of numerical solutions in room air distributlon design.
AUTHOR Awbi **H** B
BIBINF Norway, **Oslo, Norsk** VVS, Roomvent 90, Proceedings, 1315 June 1990, paper 2, 11 figs, 12 refs. #DATE **00:06:1990** in English
ABSTRACT **A** Computational Fluid Dynamics program called ARIA developed by the author and **his** colleagues is applied to predict the flow in wall jets and in rooms for isothermal and nonisothermal situations. The diffusion of two and threedimensional wall jets with and without wall obstructions was numerically predicted and the results were corroborated with measurements in **an air** jet test facility. A twodimensional solution was used to predict the isothermal **and** nonisothermal flow in a room supplied with a cold jet from a linear ceiling diffuser. The effect of downdraught from the glazed walls of a room heated with a lowlevel upward displacement system was also predicted using threedimensional solution.
KEYWORDS numerical modelling, *air* distribution, design

#NO 3712 **On** validation of computational fluid **dynamics:** procedures for **room** air **motion prediction.**
AUTHOR Baker A J, Kelso R M
BIBINF USA Preprint, ASHRAE Transactions, Vol.96, **Part** 1, 1990, 15pp, 9 **figs,** 15 refs. #DATE **00:00:1990** in English
ABSTRACT An ongoing ASHRAE research project seeks to identify and clarify key issues governing literate application of computerbased analytical methods for prediction of room **air** motion. The engineering Geld is termed "computational fluid dynamics", with the acronym **"CFD"** which is maturing rapidly, paced by **the** incredible growth of scientific computing hardware capacity. ***As*** with any emerging technology, the production of reliable predictions requires a full understanding of intrinsic details. This paper addresses issues promoting literate use of CFD for prediction of room **air** fluid/thermal flow fields.
KEYWORDS validation. **air** movement

#NO 3112 Free **convective** turbulent flow **within** the Trombe wall channel.
AUTHOR Borgers T R, Akbari H
BIBINF Solar Energy, Vol 33, No.3/4 pp253264, 1984, 11 **figs,** 19 refs. #DATE 00:00:1984 **in** English
ABSTRACT A study of free convective turbulent heat transfer between parallel plates has been made. The initial flow is assumed to remain laminar until a combination of geometry, temperature, and **flow** rate conditions reach a predefined level. At this point the modcl used in this study assumes transition and permits laminar flow to gradually develop into fully turbulent flow. Turbulent flow characteristics are predicted by a mixing length model which incorporates empirical parameters used in the literature. Using air **as** a fluid, a widerange of channel geometries, relative surface temperatures, and flow rates have been examined. Guided by the very limited available experimental data, computations were made and several correlations were developed to enable important quantities to be estimated given the channel geometry, surface temperatures and inlet **air** temperature.
KEYWORDS convection, turbulent **flow,** heat transfer, model, laminar flow, temperature

#NO 1848 The use **of** a computational method to assess the safety and quality **of ventilation** In Industrial **buildings.**
AUTHOR Broyd T W, Dean R B, Oldfield S G et al.
BIBINF I **Mech** E Conference on Heat and Fluid Flow in Nuclear and Process Plant Safety, London, 1983. C97/83. p6574. 10 figs, 1 tab, 14 refs. #DATE 00:00:1983 in English
ABSTRACT Industrial buildings, particularly those **containing** nuclear and process plant, often require high standards of ventilation in order to cope with unusual features of the operations or process which take place within the buildings. Four examples of recent studies are given in this paper: storage of **coal** in a covered stockyard, a chloralkali plant, a clean room, and the turbine hall of a **nuclear** power station. **In** each of these examples, quite detailed information **was** required about **air** flows, temperatures and gas concentrations, and it was decided to use the twoequation model of turbulence to help predict these variables. This is solved with equations for momentum and continuity by finite differences. It is concluded that complex computer programs of this kind can provide valuable assistance in support of the more traditional hand calculations. However, careful engineering judgement must be exercised in the use of the programs and in the interpretation of the results.
KEYWORDS mathematical modelling, industrial building, natural ventilation, mechanical ventilation

#NO 1903 The **use** of computational modeling techniques **for** cleanroom design.
AUTHOR Broyd T W, Deaves D M, Oldfield S G
BIBINF Semicon/Europa 1983 Semiconductor Processing and Equipment Symposium, Zurich, Switzerland, March 1983. p324347. 23 **figs,** 1 tab, 4 refs. #DATE 00:03:1983 in English
ABSTRACT This paper presents the results of a computational and experimental study to assess the possible benefits of using mathematical modeling techniques for cleanroom design. A twodimensional modeling visualization has been used for each of the cases studied. This has the advantage that its relative cheapness allows more design variations to be analyzed than would be possible with a threedimensional method. Fullscale measurements were also taken in the cleanroom which was the subject of the modeling exercise. The predictions of the computational model were found to be in good agreement with the measurements taken.
KEYWORDS mathematical modelling, **air** movement

#NO 3839 Flow **3 D bodyfitted coordinates.**
AUTHOR Burns A D, Wilkes N S, Jones I P, Kightley J R
BIBINF UK, UKAEA Harwell, Computer Science and Systems Division, June 1986, 9pp, 9 ***figs,*** 21 refs. #DATE 00:06:1986 in English
ABSTRACT This paper discusses the extension of the FLOW3D code to handle flows in complex geometries using BodyFitted Coordinates. The code uses a new algorithm, that of Rhie and Chow, to deal with the problems in a simple and efficient manner using nonstaggered grids. This technique is briefly explained and its implementation in the code outlined. Other enhancements to the code are also described. Finally, the code is applied to a number of test problems. These are the laminar flow in a curved duct, for which accurate fmite element results are available, and the turbulent flow in a rectangular **box.** A variety of grid distributions are used in the latter case and the rcsults are compared with FLOW3D results using the standard algorithm. The results are in good agreement with the benchmark cases and they clearly demonstrate the ability of the code to handle problems in complex geometries.
KEYWORDS air flow

#NO **3842** Steady, twodimensional, natural convectlon in rectangular enclosures with **differently** heated walls.
AUTHOR Chen K S, Ho J R, Humphrey J A C

BIBINF USA, Transactions of the **ASME,** Vol 109, May 1987, **pp400406, 12** figs, 1 tab, refs. #DATE 00:05:1987 in English

ABSTRACI' Numerical results are presented for steady natural convection in twodimensional rectangular enclosures in which the side walls, top wall and bottom wall are at uniform temperatures 0, 0, and 0, respectively, and 000. Rayleigh numbers ranging from 10 to the power of **4** to 10 to the power of 7 and aspect ratios of 1 and 1.5 were investigated, The top wall was modelled **as** an impermeable rigid surface or an impermeable freemoving boundary. The calculations reveal two flow regions. In the upper part of the enclosure two large countcrrotating cells appear, separated by a descending plume of fluid. Near the bottom wall the flow is almost motionless and stably stratified. The temperature in the central portion of the enclosure is almost uniform due to mixing by the recirculating cells. A temperatwe inversion occurs near the top wall and **is** particularly noticeable at high Rayleigh numbers. At high Rayleigh numbers the flow breaks up into smaller cells. The result is that each main recirculation region develops a secondary counterrotating eddy within it. The condition of a free surface **as** the top wall boundary condition significantly affccts the circulation and heat transfer throughout the **flow** domain. Numerical experiments reveal the extent to which the floor field in the enclosure is affected by an asymmetric specification of sidewall temperature boundary conditions.

KEYWORDS convection, wall, air flow

#NO 3846 Experlments and 2D approximated computations of 3D air movement heat and concentration transfer In a room.

AUTHOR Chen **Q,** Van der Kooi J

BIBINF Sweden, Proceedings of the International **Con**ference on Air Distribution in Ventilated Spaces, Roomvent 87, Vol 40, 1012 June 1987, pp114, 5 figs, 1 tab, 7 refs. #DATE 00:06:1987 in English

ABSTRACT A few approximated methods were used to predict 3D mixed **air** movement of natural and forced **con**vection with concentration transfer in a room by a **2D** computer code CHAMPION SGE. The computation method involved the solution, in finitedomain form, of two dimensional equations for the conservation of mass momentum, energy, concentration, turbulence energy and dissipation rate, with wall function expressions for solid boundary conditions. Corresponding measurements of temperature distributions and flow patterns were used for comparison. The following **cases** were investigated: natural convection in the room; the room with solar radiation through the **window** and with cold air supply, also with a CO2 **gas** source. The approximated methods were: treating the heat transfer through **the** walls in the Zdirection as thermal sourccs or sinks for the finitedomainequation; **con**centrating the heat transfer through the walls in X and Y direction; and using equivalent inlet temperatures.

KEYWORDS air flow

#NO 2979 Accuracy a program for comblned problems of energy analysis indoor alrflow and air quallty.

AUTHOR Chen **Q,** Van der **Kooi** J

BIBINF Preprint: Ashrae Transactions, Vol94, Pt 2, 1988, 2 figs, 3 tabs, 19 refs. #DATE 00:00:1988 in English

ABSTRACI' A new userfriendly, personal computer program, ACCURACY, **has** been constructed for energy analysis, room **air** temperature and contamination field predictions. The program is **a** coupling of a **cooling** load program and **an airflow** program. It is based on the room energy balance method and uses Ztransfer functions and window energy balance equations for heat transfer through enclosures. The computer program also involves the solution, in finitedomain form, of threedimensional, transient equations for the conservation of energy and **con**tamination concentration **using** airflow patterns, which are precalculated from dedicated computer programs, such as PHOENICS (3D) and CHAMPION **SGE** (2D). The influence of room air supply system on the temperature distribution and indoor **air** quality **and** subsequently on the energy consumption can be calculated by ACCURACY. The agreement between measured and simulated results by ACCURACY **is** good.

KEYWORDS air flow, air quality, energy audit, computer

#NO 3845 Measurements and computatlons of ventilation efficiency and temperature efflciency In a ventilated room.

AUTHOR Chcn **Q, Van** der Kooi J, Meyers A.

BIBINF UK, Energy and Buildings, No.12, 1988, pp8599, 12 **figs,** 3 tabs, 7 refs. #DATE **00:00:1988** in English

ABSTRACI' In order to improve the indoor **air** quality **in** a room and to save energy, the air movement **and** contamination distributions in the room with ventilation have been studied experimentally and numerically. The experiment is carried out in a fullscale climate room with different air supply systems, heat gains from the venetian blinds and ventilation rates. The measurements concern room **air**flow patterns and **air** temperature, velocity and contamination concentration fields, etc. The airflow computer program PHOENICS and **the** cooling load program ACCURACY have been applied for the numerical simulations. **PHOENICS** solves the conservation equations of **air** mass, momentum, energy, concentration, kinetic energy and dissipation rate of kinetic energy. ACCURACY, which considers thc influence of room **air** temperature distributions, is employed for the determination of cooling load, wall surface tcmperatures and convective heat transfer on room enclosure surfaces, These are the boundary conditions required by **PHOENICS.** The agreements between the computations and the measurements arc good. The ventilation efficiency and temperature efficiency which are used for evaluation of indoor **air** quality and en**ergy** consumption are reported for each **case.** Additional application of these computations to annual energy analysis is also discussed.

KEYWORDS air flow, ventilation efficiency

#NO 3652 Indoor airflow, air quallty and energy consumptlon of buildings

AUTHOR Chen **Q**

BIBINF Netherlands, Technical University Delft, 1989. **#DATE** 00:00:1989 in English
ABSTRACT The aim is to study airflow, contaminant concentration distributions and air temperature fields in an airconditioned room with different kinds of air supply and air exhaust system and to predict the heat extracted from and supplied to the room, and consequently the energy consumption of these systems. This will allow the development of reliable methods for estimating the influence of indoor airflow on the comfort and energy consumption of buildings, for controlling the pollutant dispersion, and for increasing the efficiency of energy utilization. The research is principally carried out by numerical techniques, in particular a twodimensional airflow program and a threedimensional airflow program are used, and an airconditioning load program is constructed. The second aim of this research is to compare the computational results with experiments. Therefore, corresponding measurements are performed.
KEYWORDS airflow, air quality, energy consumption

#NO 3287 Convective heat transfer In room8 with mixed convectlon.
AUTHOR Chen Q, Meyers C A, Van der Kooi J
BIBINF Preprint: International Seminar on Indoor Air Flow Patterns held at Laboratory of Thermodynamics, University of Liege, Belgium, 9th February 1989, 14pp, 11 figs, 4 tabs, 10 refs. #DATE 00:02:1989 in English
ABSTRACT Experiments have been carried out for obtaining the convective heat transfer coefficientsin the enclosure surfaces of a room with mixed convection. It has been found that the convective heat transfer coefficient is about 4.0 W/m2K for the ceiling and 4.7 for the floor with unstable flow and 4.3 for the floor with stable flow. Due to the measuring technical problems, the measuring errors are too large. We may also conclude that the convective heat transfer coefficient is about 10.7 W/m2K for the venetian blind surface facing to the room and about 4.5 for the venetian blind surface facing to the glass pane of the window. These results only present a rough indication of the heat transfer in a window with venetian blinds. Those measured convective heat transfer coefficientshave been applied in the numerical simulation of indoor airflow. This has achieved acceptable agreement between the computed and measured velocity and temperature distributions of room air.
KEYWORDS convection, heat transfer, blind

#NO 3710 Predlctlon of buoyant, turbulent flow by a low Reynolds number ke model.
AUTHOR Chen Q, Moser A, Huber A
BIBINF USA Preprint, ASHRAE Transactions, Vol.96, Part 1, 1990, 10 pp, 11 figs, 1 tab, 25 refs. #DATE 00:00:1990 in English
ABSTRACT A lowReynolds number ke model of turbulence was used for the prediction of natural convection flow in cavities with Rayleigh numbers on the order of 10. The Boussinesq approximation was used for buoyancy, and the buoyancy production terms in the k and e equations were also studied. The results indicate that the computed velocity and temperature profiles and convective heat exchanges by the model are in rather good agreement with the measurements. The influence of the buoyancy production is small on velocity and temperature profiles but is considerably large on the kinetic energy profiles. For the indoor airflow computation, use of the lowReynolds number model with buoyancy production terms is recommended so that correct indoor air velocity fields, air temperature distributions. convective heat transfer coefficients, and comfort parameters *can* be obtained.
KEYWORDS prediction, turbulent flow, model

#NO 4154 A methodology for Indoor airflow computatlons and energy analysls for a dlsplacement ventllatlon system.
AUTHOR Chen Q, Van der Kooi J
BIBINF Switzerland, Swiss Fed Inst of Technology, ETH Zurich, paper to be published in Energy and Buildings, 1990, 13 pp, 8 figs, 2 tabs, 34 refs. #DATE 00:00:1990 in English
ABSTRACT This article presents a methodology for the computation of indoor airflow, air quality, space load and energy consumption of a room with a displacement ventilation system. Since airflow and transient heat transfer in the room are interrelated, the indoor airflow and the space load of the room must be predicted simultaneously. In order to reduce the computing costs, a simplified method has been introduced for the predictions. According to the present state of the art, the kE turbulence model is suggested for the indoor airflow computations within a room. The air temperature distributions of the room are used in a cooling load program for space load calculations. This is very important for a room with a displacement ventilation system. An optimized algorithm is applied for the estimation of the energy consumption of the room. Finally, an application example is presented. The results indicate that, in a room with the displacement system, the indoor air quality is much better than with a wellmixed system, and the energy savings are significant.
KEYWORDS displacement ventilation, calculation techniques

#NO 3719 Energy analysis of bulldlngs with dlfferent air supply and exhaust systems.
AUTHOR Chen Q, Hoornstra T G, Van der Kooi J
BIBINF USA, Preprint, ASHRAE Transactions, Vol.96, Part 1, 1990, 13pp, 11 figs 11 tabs, 16 refs. #DATE 00:00:1990 in English
ABSTRACT Research was carried out in order to study the influence on energy consumption of different kinds of air supply and exhaust systems in a room. Based on the air temperature distribution of the room with a displacement ventilation system and a wellmixed system, which were calculated by an airflow program, the cooling load program ACCURACY and the energy analysis program ENERK were used for the calculation of the space load and the annual energy consumption of the room. The results showed that the air temperature distributions in the room are very importaut in the prediction of room energy consumption. For a variable airvolume system, the energy

required by the chiller and the ventilator with the displacement ventilation system is 26% smaller than that with the wellmixed system. The air displacement system is recommended for practical applications for saving energy and obtaining better indoor air quality.
KEYWORDS energy consumption, displacement ventilation, mechanical ventilation

#NO 3851 Natural convection and air flow pattern in a partitioned room with turbulent flow.
AUTHOR Haghighat **F,** Jiang Z, **Wang** J *C* **Y**
BIBINF USA, Preprint, Ashrae Transactions, Vol 95, Part 2,1989, llpp, 11figs, 1tab, refs. #DATE 00:00:1989 **in** English
ABSTRACT Recent studies have emphasized the importance of the interzone *air* movement in **a** building **and** demonstrated the need for better understanding of this movement in any attempt to predict the ventilation efficiency or thermal performance of the building. This paper discusses the use of the ke twoequation turbulence model to simulate natural convection of **high** Rayleigh number in a partitioned enclosure for a few *cases.* The **airflow** in all cases is considered to be three dimensional owing to the asymmetry of the room configuration. The predictions of the model are compared to available experimental and theoretical results; good agreement is obtained. The study also discusses the effect of door height and location on the pattern of **airflow** and temperature. Results indicate that the flow pattern is quite sensitive to the variations of door height and location, while the convective heat transfer rate is only sensitive to variation of door height.
KEYWORDS air flow, convection

#NO 2751 The contaminant distribution in a ventilated room with different air terminal devices.
AUTHOR Heiselberg **P,** Nielsen P V
BIBINF Institute of Building Technology and Structural Engineering, Aalborg University, **Denmark,** August 1987, **14p,** 9 **figs,** 8 refs. **#DATE** 00:08:1987 **in** English
ABSTRACT The room ventilation is investigated for three different air terminal devices under isothermal conditions. Velocity distribution in the occupied zone is measured for each air terminal device at different air exchange rates. The maximum air exchange rate is determined on the basis of both the throw of the jets and the comfort requirements applied to measured air velocities in the occupied **zone.** Normalised concentration distribution in the test room is determined along **a** vertical line through the middle of the room as a function of the *air* exchange rate and the density of the tracer **gas.** The relative ventilation efficiency,based on the room avcragc concentration is **also** determined **as** a function of the *air* exchange rate and the density of the tracer **gas.** The influence from the position of the return opening on the relative ventilation efficiency is found for one air terminal device.
KEYWORDS pollutant, air change rate, tracer gas, ventilation efficiency,instrument

#NO 3146 Flow conditions in a mechanically ventilated room with a convective beat source.
AUTHOR Heiselberg **P,** Nielsen P V
BIBINF in: "Effective Ventilation", 9th AIVC Conference, Gent, Belgium, 1215 September, 1988. #DATE 00:09:1988 in English
ABSTRACT The ventilation of a test room (LxWxH = **5.4x3.6x2.4m)** with a wall mounted heat source is investigated for two different *air* terminal devices. The properties of each air terminal device are described by measuring the velocity decay of the primary **wall** jet below the **ceiling.** The velocity distribution in **the** plume above the heat source has **been** measured at different heat loads **as** a function of the distance to the **wall** and the distance to the heat source. The measurements have led to an estimate of the **maximum** velocity in the plume and of the volume flow rate **as a** function of the heat load and the distance to the heat **source.** In order to find the influence of the convective heat **source on** the flow conditions **in** the room, the velocity distribution in the occupied **zone** and the normalized concentration distribution along a vertical line through the middle of the room has been determined **as** a function of the specific flow rate and the heat load. The convective heat source is found to have significant influence on the flow conditions in the room. This paper shows lower velocities in the occupied zone and a more uniform concentration distribution in the room.
KEYWORDS air flow, mechanical ventilation, convection heating, mathematical modelling

Calculation of the air flow pattern in a proposed new European test chamber for radiators.
AUTHOR Huber **A,** Moser **A,** Chen Q
BIBINF UK, AIVC proceedings of the 11th AIVC Conference "Ventilation System Performance", Belgirate, Italy, 18-21September, pp329-345.

#NO 2145 Evaluation of ventilation systems through three **dimensional numerical cornputation.**
AUTHOR Ishizu Y
BIBINF Trans SHASE, No.30, February 1986, p17, 15 **figs,** 3 refs. #DATE 00:02:1986 in English
ABSTRACT To make **an** evaluation of ventilation systems, **numerical** computation was carried out for three dimensional, isothermal, and turbulent flow schemes. It was found that there exists **an** optimum position for an inlet in tclation to an outlet whereby the most effective ventilation **can** be attained. In addition, **similar** to the results for the two dimensional computation, the slope of the concentration decay is virtually constant and independent of the position in the room, so the **mixing** factor derived from this slope *can* be used **as an** index of the ventilation efficiency. Further, three dimensional computation **seems to** be necessary for a quantitative estimation of the mixing factor.
KEYWORDS mathematical modelling, ventilation strategy, ventilation efficiency.

#NO 3462 Modelling heat, moisture and contaminant transport In buildings: toward a new generation software.

AUTHOR Kerestccioglu H, Swami M, Fairey P, Lixing G, Chandra S
BIBINF USA, Florida Solar Energy Center, FSECPF16589, 1989, 8pp, 9 figs, 11 refs. #DATE 00:00:1989 in English
ABSTRACT This paper describes a general purpose software, Florida Software for Engineering Calculations, that is capable of solving various transport equations used in building science (eg combined heat and moisture transfer, fluid flow, contaminant dispersion equations, etc). The governing equations are solved by finite element methods. General capabilities and an overview of the software structure are given. Results arc presented for several types of combined heat and moisture transfer simulationsL 1) in buildings; 2) in the presence of natural convection; 3) in attics 4) in a typical wall.
KEYWORDS contaminant, computer, moisture, attic, wall

#NO 2161 Model tests of ventilation effectiveness and air distribution literature survey.
AUTHOR Klobut K.
BIBINF Report C:13, Espoo, Finland: Helsinki University of Technology, Laboratory of Heating, Ventilation and Air Conditioning, Indoor Climate Project, 1985. 114p, 34 figs, 8 tabs, 84 refs. #DATE 00:00:1985 in English .
ABSTRACT A survey of mathematical models of air flow and of ventilation efficiency. Measuring equipment for laboratory experiments is described and the limitations and potential uses of the models are discussed.
KEYWORDS mathematical modelling, air flow, ventilation efficiency

#NO 3843 Numerical simulation of air movement and temperature field In a room with cold and hot window surface.
AUTHOR Kooi J, Chen Q
BIBINF Netherlands Proceeding of Euromech Colloquium 207 in Natural Convection, Delft, 79 April, 1986, 2pp, 6 figs, 5 refs. #DATE 00:04:1986 in English
ABSTRACT The air velocity and temperature characteristics of room are important to the comfort and well being of occupants. Computations were carried out to predict air flows and heat transfer in a room with natural convection by applying the cornputer codes CHAMPION SGE and PHOENICS81. The computational results of the temperature field and velocity field are compared with corresponding experiments.
KEYWORDS air movement, window, temperature

#NO 3850 Thermal performance of a room with air displacement ventilation system.
AUTHOR Kooi J, Chen Q
BIBINF Yugoslavia, Proceedings of the Second World Congress of Heating, Ventilating, Refrigerating and Air Conditioning, CLIMA 2000, Sarajevo, 27 August1 September 1989, 2 figs, 6 tabs, 4 refs. #DATE 00:00:1989 in English
ABSTRACT The usual cooling load programs can only be used for rooms with a uniform temperature distribution. With a new program ACCURACY it is also possible to calculate situations with a vertical temperature stratification. With this program, the hourbyhour cooling/heating loads have been calculated for a room with a displacement ventilation system (with temperature stratification) and a wellmixed system (without temperature stratification). With an energy calculation program, ENERK, the yearly energy consumption of both systems has been calculated and mutually compared, For the same inlet air temperature, the energy costs of the displacement ventilation system are about 16% lower in comparison with the wellmixed system in the situation considered.
KEYWORDS thermal performance, displacement ventilation

#NO 3911 A numerical method for calculating indoor air flows using a turbulence model.
AUTHOR Kurabuchi T, Fang J B, Grot R A
BIBINF USA, National Institute of Standards and Technology, NIST, R894211, January 1990, 116 pp. #DATE 00:01:1990 in English
ABSTRACT This report describes a numerical method based on a finite difference technique for simulating indoor airflows in a building using a KE turbulence method. The model treats three dimensional nonisothermal turbulent flows using the Boussinesq approximation for buoyancy. It solves the resulting nonlinear system of momentum, energy and turbulence equations by an explicit time marching technique to obtain a solution for either a steady state or transient flow. An upwind/central combination scheme with arbitrary specification for the switching parameter is used to approximate the convective terms. This switching parameter can bc specified at each point in the flow regime allowing for different strategies in different flow regions. The switching technique includes both the central and hybrid schemes found in the literature. A pressure relaxation method is used to satisfy the Poisson equation for continuity. The model handles a variety of flow, pressure, temperature and heat flux boundary conditions including prescribed inflows, outflows by either prescribing the flow or pressure, wall boundary conditions together with heat flux and temperature and/or heat transfer coefficients specified on the boundary. Volumetric heat sources are also included. The model has the ability of handling an arbitrary number of obstacles in the flow region. This permits the modelling of the effect of furniture and partitions on the flow field and also provides a means for modelling multiroom airflows. The predicted airflows can be used in a companion computer model for predicting the threedimcnsional dispersion of contaminants in a building. Isothermal simulations seem to converge in approximately 10,000 iterations and nonisothcrmal simulations in approximately 30,000 iterations. Several ideal and practical applications of the model are presented and the results of the simulations are compared with existing experimental data contained in the literature.
KEYWORDS air flow, contaminant, indoor air quality, mathematical modelling

#NO 3866 A modified form of the ke model for predicting wall turbulence
AUTHOR Lam **C K** G, Bremhorst **K**
BIBINF USA, Transactions **of** the **ASME,** Vol103, S e p tember 1981, **pp456460, 4 figs,** refs. #DATE 00:09:1981 in English
ABSTRACT The **high** Reynolds number form of the ke model is extended and tested by application to fully developed pipe flow. It is established that the model is valid throughout the fully turbulent, semilaminar and laminar regions of the flow. Unlike many previously proposed forms of the ke model, the present form does not have to be used in conjunction with empirical **wall** function formulas **and** does not include additional terms in the k and e equations. Comparison between predicted and measured dissipation rate in the important wall region is **also** possible.
KEYWORDS air flow, modelling

#NO 4129 A numerical study of the air movement and temperatures in large atria and sunspaces.
AUTHOR Lemaire **A D**
BIBINF Norway, Oslo, Norsk W S , Roomvent 90, proceedings, **1315 June 1990, paper 5, 15** pp, **11 figs, 4** refs. #DATE 00:00:1990 in English
ABSTRACT The air movements and temperatures in two atria and **two** sunspaces under winter conditions have been simulated numerically. Aim of the study **was** the prediction of the thermal comfort in the occupied zone of large glazed spaces with different heights and with different types of heating systems. The spaces were rectangular shaped with a base of **24** by 24 meters and a height of respectively 32 and ***64*** meters. The threedimensional, turbulent airflows have been simulated by the computer code **WISH3D (** a **TNO** development**),** using the kE turbulence model and the finite volume method. Transient calculations were applied to attain steady state solutions and to detect unstable **flows.** To prevent expensive computer costs only **a** limited number of cases have been studied with a reduced accuracy, Nevertheless, useful information regarding the ***air*** movements **and** temperatures in large **glazed** spaces was obtained from the results.
KEYWORDS atrium, numerical modelling, sunspace, air movement, temperature

#NO 3291 Air Infiltration Calculation Techniques An Applications Guide.
AUTHOR Liddament M W
BIBINF UK, AIVC Applications Guide **1,1986** #DATE **00:00:1986**
ABSTRACT A Loose leaf handbook divided into ***six*** chapters covering empirical and theoretical calculation techniques. algorithms, references and **glossary** of terms.
KEYWORDS calculation techniques

#NO 3862 The ASTEC code: an algorithm for solving thermal hydraulic equations in complex geometries.
AUTHOR Lonsdale **R D**
BIBINF UK, ***AEA,*** Dounreay Nuclear Power Development Establishment, 1988,8pp, 11 figs, 12 refs. #DATE 00:00:1988 in English
ABSTRACT By applying **a** finite volume approach to **a** finite element mesh, the ASTEC computer code allows threedimensional incompressible fluid flow and heat transfer in complex geometries to be simulated realistically, without making excessive demands **on** computing resources. The methods used in the code are described, and examples of the application of the code are presented.
KEYWORDS ***air*** flow, computer, mathematical modelling

#NO 1526 Computer analysis of building Ventilation and heating problems.
AUTHOR Markatos **N C.**
BIBINF 2nd International **PLEA** Conference, Crete, 28 June1 July 1983. Passive and **Low** Energy Architecture, **edited** by Simos **Yannas.** Oxford:Pergamon,1983. **p667675,** 14 **figs,** 10 refs. #DATE 00:07:1983 **in** English
ABSTRACT This paper presents a mathematical model, implemented in a general computer code, that ***can*** provide detailed information on the velocity and temperature fields prevailing in threedimensional buildings of **any** geometrical complexity, for a given ventilation and heating arrangement. The model is based on the partial differential equations governing **flow** and heat transfer in large enclosures, containing heat sources. Turbulent flow is simulated and account is taken of buoyancy effects on both mean and fluctuating motions. Two cases are **considered**, to demonstrate the capabilities of the present model, both referring to actual existing television studios. The program calculates the velocity and temperature fields throughout the threedimensional configurations, and the results are presented in the form of velocity vectors and temperature contours. The same procedure is applicable to **any** problem of heating, cooling, insulation and ventilation of buildings! predicting within practical resources the thermal and fluid dynamics behaviour of the relevant systems.
KEYWORDS mathematical modelling, simulation, **air** movement, ventilation, heating, heat transfer

#NO 2190 Numerical analysis of room air distribution by the finite element method.
AUTHOR Matsumoto **H,** Hasegawa F, Utsumi Y.
BIBINF Journal **of** Architecture, Planning and Environmental Engineering (Transactions of AIJ), No 352,1985, **p3139, 27 figs, 2** tabs, 32 refs. #DATE **00:00:1985** in Japanese
ABSTRACT This paper describes the numerical analysis of room **air** distribution by the finite element method which can easily deal with **any domain,** the boundary **conditions** and so on. The twoequation model of turbulence is applied to the governing equations of room **air,** and the discretization of the basic equations is formulated by **the** penalty finite element method. The finite clement equations are solved by the partitioning method that the equations are partitioned into the momentum equation of the mean flow and the transport equations for turbulence kinetic **energy** and that for turbulence energy dissipation

rate, and the modified Newton method is employed in the iterative procedure. The accuracy and the stability of the scheme by the influence of the penalty parameter are examined for the two dimensional Poiseuille flow. Though the accuracy of the solutions is improved as the penalty parameter is increased, the large parameter makes the condition of the coefficient matrix ill and the numerical convergence is hard to be obtained for the computer. In this computational experiment the scheme has good accuracy. At last the numerical example of the three dimensional room model is carried out and the solutions are confirmed to be fully sufficient. be fully sufficient. As a result, the finite element method is effective for the prediction of the room air distribution.
KEYWORDS air flow, turbulence

#NO 2717 Numerical analysis of airflow and pollution in buildings.
AUTHOR Matsumoto H, Hasegawa F, Utsumi Y
BIBINF in: Third International Congress on Building Energy Management, III Ventilation, air movement and air quality: field measurement and energy auditing, held in Lausanne, Switzerland, September 28 October 2,1987, p128135, 5 figs, 11refs. #DATE 00:09:1987 in English
ABSTRACT In an air conditioning system, it is necessary to remove or to supply heat in order to maintain a comfortable temperature level, but it is also necessary to supply the room with a proper amount of fresh air. Therefore the indoor airflow is an important factor in the investigation of thermal comfort, ventilation efficiency, air pollution and cncrgy conservation. Model experiments and numerical analysis are known to predict indoor airflow and distribution of air pollutants. In the modcl experiments, the accuracy of prediction is affected by the level of the measuring techniques. Even though accurate and highly responsive instruments, such as an infrared gas detector or a flame ionization detector (FID), are used, it is difficult to analyse the detailed structure of the turbulent diffusion. This paper describes a numerical method of airflow and air pollution in buildings by the finite element method, and shows the mechanisms for the diffusion of air pollutants by cornputer simulations.
KEYWORDS numerical modelling, air flow, pollution, thermal comfort, ventilation efficiency, energy conservation, instrumentation, computer, simulation

#NO 838 Room flow tests in a reduced scale.
AUTHOR Moog W.
BIBINF ASHRAE trans. no.1, 1981, p.11621185.1 tab. 2 figs. 15 refs. #DATE 01:01:1981 in English
ABSTRACT The study concerns the problems and prediction of room flow in airconditioning. It is shown how difficult it is to form mathematical models, especially of the threedimensional flow field occurring in practice. After basic definitions, an explanation of the influence of different air flow systems on the structure of room flow is given. The microstructure of the flow is so complex that a precise mathematical model formation seems to be impossible. Measurements on isothermal and models(reduced from the original)supply reliable threedimensional results. Similaritytheoretical studies indicate that model tests are principally a useful *means* of increasing the dimensional E r - tainty, but in order to exactly reproduce the thrcedimensional room flow, the test in the scale 1:1 remains essential.
KEYWORDS air conditioning, mathematical modelling, empirical mode4

#NO 3549 Trends la airflow deslgn and management, contributions by TEA Annex 20.
AUTHOR Moser A
BIBINF **in:** UK, 10th AIVC Conference Proceedings Volume 1, held Espoo, Finland, 2528 September 1989, published February 1990, pp4562, 4 figs, 12 refs #DATE 00:02:1990 in English
ABSTRACT What does the designer of a future energyefficient building ask of the air flow specialist? Static predictions of air flow patterns and optimization of thermal comfort and indoor air quality at design conditions will not be enough for him. The paper suggests that timedependent air flow simulation is imperative to respond to tomorrows design needs. Different physical time scales for air flow patterns in spaces will be discussed. Heat capacity by components, different types of heat transfer, varying occupancy, control inputs etc. give rise to disparate time scales. The trend toward occupant controlled ventilation will continue. Air Flow will interactively be adjusted to changing needs in each room. The IEA Annex 20 examines tools to predict steady air flow patterns within buildings. The dynamic management of air flows will require new methods that build on Annex 20 work.
KEYWORDS air flow, occupant control

#NO 2623 Three dimensional numerical slmulatlon of turbulent airflow ln a ventllated room by means of two equation model.
AUTHOR Murakami **S,** Kato S, Suyama **Y**
BIBINF Preprint. Ashrae Trans, Vol 93, Part 2, No 3091, 1987, 17p, 16 figs. #DATE 00:00:1987 in English
ABSTRACT Turbulent recirculating flows in many types of ventilated rooms are numerically simulated threedimensionally by means of a kE twoequation turbulence model. The results obtained from numerical simulation are compared with those given by modcl experiments concerned with velocity and diffusion field. The correspondence between simulations and experiments is fairly good. Proposed threedimcnsional numerical simulation by means of a kE twoequation model cnablcs the prediction of turbulent flow distribution in a ventilated room with sufficicnt accuracy.
KEYWORDS turbulent air flow, prediction, mathematical modelling

#NO 2976 Numerical and experimental study on turbulent diffusion flelds in conventional flowtype clean rooms.
AUTHOR Murakami **S,** Kato S, Suyama Y
BIBINF Preprint: Ashae Transactions, Vol 94, Pt 2,1988, 23 pp, 19 figs, 3 tabs, 5 refs. #DATE 00:00:1988 in English

ABSTRACT Turbulent flow fields of velocity and diffusion in several types of conventional dean rooms are precisely analyzed both by model experiment and by numerical simulation based on the K twoequation turbulence model. The detailed analyses of contaminant diffusion by simulation makes it possible to comprehend clearly the structures of velocity and diffusion fields in clean rooms. The flow fields in such rooms, as analyzed here, are mainly characterised by the inflow jet and the rising streams around it. The combination of one jet and rising streams forms a "flow unit". The total velocity field and the resulting diffusion field of contaminant in a room are well modeled as serial combinations of these "flow units".
KEYWORDS turbulence, cleanroom, numerical modelling, pollutant

#NO 3853 Numerical study on diffusion field as affected by arrangement of supply and exhaust openings In conventional flow type clean room.
AUTHOR Murakami S, Kato S, Suyama Y
BIBINF USA, Preprint, Ashrae Transactions, Vol 95, Part 2, 1989, 15 pp, 12 figs, 6 tabs, refs, #DATE 00:00:1989 in English
ABSTRACT Room air distribution is greatly affected by the arrangement of supply outlets and, possibly, exhaust inlets. Influence of those arrangements on the flow fields is studied here by numerical simulation based on the ke twoequation turbulence model. Room airflows in several types of conventionalflowtype clean rooms are analyzed from this point of view. The flow fields in such rooms as analyzed here are well modelled as serial combinations of "flow units", each of which is composed of one supply jet and the rising streams around it. When the number of supply outlets is decreased, the flow units corresponding to the eliminated supply outlets vanish and the remaining flow units expand. A change in arrangement or in the number of exhaust inlets hardly affects the entire flow field; however, such changes often have a large influence on the contaminant diffusion field.
KEYWORDS air flow, cleanroom

#NO 3852 3D numerical simulation of turbulent air flow In and around buildings based on the ke model with generalized curvilinear coordinates.
AUTHOR Murakami S, Kato S, Ishida Y
BIBINF USA, Preprint, Ashrae Transactions, Vo1.95, Part 2,1989, 28pp, 23 figs, 7 tabs, refs. #DATE 00:00:1989 in English
ABSTRACT The air distribution in and around a building with a complicated congfiguration is well simulated by the finite difference method based on generalized curvilinear coordinates. This paper follows preceding studies which were based on ordinary Cartesian coordinates. Numerical simulations of room airflow by the present method using the ke model based on curvilinear coordinates are conducted. Its validity and feasibility for application to engineering problems arc confirmed by comparing simulation results with the experimental results.
KEYWORDS air flow, numerical modelling

#NO 3331 Numerical and experimental study on room airflow 3D predictions using the ke turbulence model.
AUTHOR Murakami S, Kato S
BIBINF Building and Environment, Vol 24, No 1,1989, pp8597, 14 figs, 3 tabs, 16 refs. #DATE 00:00:1989 in English
ABSTRACT Accurate prediction of velocity and temperature distributions in a room is indispensable for designing high quality air conditioning systems. This paper is concerned with the feasibility and validity of numerical simulation of room airflow. Turbulent recirculating flows in many types of ventilated rooms were numerically simulated in threedimensions using the ke two equation turbulence model. The results obtained from the numerical simulation are compared with those given by model experiments concerned with velocity and diffusion fields. The correspondence between simulations and experiments is fairly good. It may be concluded that 3D numerical simulations using the ke two equation model can predict turbulent recirculating flows in a ventilated room with sufficient accuracy from the viewpoint of engineering applications.
KEYWORDS air flow, turbulence, numerical modelling

#NO 3338 Three dimensional numerical simulation of turbulent flow around buildings using the ke turbulence model.
AUTHOR Murakami S, Mochida A
BIBINF Building and Environment, Vol 24, No 1,1989, pp5164, 25 figs, 16 refs. #DATE 00:00:1989 in English
ABSTRACT Threedimensional numerical simulations of air flow around a cubic model and building complex using the ke two equation turbulence model are presented in this paper. Several cases of numerical simulation of airflow around a cubic model are carried out to estimate the influences of a mesh dividing system and boundary conditions on simulated results. The accuracy of these simulations is examined by comparing the predicted results with wind tunnel experiments conducted by the authors. It is confirmed that numerical simulations by means of the ke model reproduce the velocity and pressure fields well when using fine mesh resolution around the model. In the latter half of this paper, the numerical method is applied in order to predict the flow field around a building complex under construction at present. The applicability of the numerical method in practical situations is demonstrated.
KEYWORDS numerical modelling, turbulent airflow

Numerical and experimental study on flow and diffusion field In room.
AUTHOR Murakami S, Kato S
BIBINF UK, AIVC, proceedings of 11th AIVC Conference, "Ventilation System Performance", Belgirate, Italy, 18-21 September 1990,121-156.

#NO 2752 Air distribution in rooms with ceiling mounted obstacles and three dimensional Isothermal flow.
AUTHOR Nielsen P V, Evensen L, Grabau P, et al

BIBINF Institute of Building Technology and Structural Engineering, Aalborg University, Denmark, August 1987, 11p, 8 figs, 11 refs. #DATE 00:08:1987 in English
ABSTRACT This paper presents results obtained with threedimensional, isothermal flows in a model room having a circular supply opening (nozzle) located in the end wall close to the ceiling in the symmetry plane of the model. The first part of the paper deals with nondeflected flow and demonstrates the influence of the Reynolds number and the influence of ceilingmounted obstacles on the penetration depth in long rooms. Some examples of velocity distribution in the wall jet and flow in the occupied zone are then demonstrated. The second part of the paper determines the critical dimensions giving a deflection of the supply jet into the lower part of the room (occupied zone),
KEYWORDS air movement, model, ceiling, supply vent

#NO 2753 Measurements on buoyant wall jet flows in alr conditioned rooms.
AUTHOR Nielsen P V, Moller A T A
BIBINF Institute of Building Technology and Structural Engineering, Aalborg University, Denmark, August 1987, 12p, 7 figs, 11 refs. #DATE 00:08:1987 in English
ABSTRACT Sidewallmounted diffusers placed in the vicinity of the ceiling in a ventilated room will often generate a flow of the wall jet type. The jet follows the ceiling, entrains air from the occupied zone and generates a recirculating flow in the whole room. This paper will deal with the flow in the ceiling region. The wall jet flow is especially influenced by diffuser design and surrounding details such as distance to the ceiling and the ceiling structure. The flow is lesser influenced by other parameters in the room such as length, width, height and furnishings. It is important to study the conditions and locations where the flow can be described as a wall jet. This description is useful when different diffusers are compared, and it is the background for calculation of "throw" and "penetration depth. It is also convenient to use the wall jet description of inlet conditions in computer predicted flow in rooms. This description makes it possible to ignore details at the diffuser as e.g. vanes which means reduction of computer storage and increased computation speed.
KEYWORDS air conditioning, air movement, ceiling, wall, air flow

#NO 3913 Numerlcal heat transfer.
AUTHOR Patankar S V
BIBINF USA, New York, Hemisphere, 1980, 197pp. #DATE 00:00:1980 in English
ABSTRACT Primarily aimed at developing a general method of prediction for heat and mass transfer, fluid flow, and related processes. Among the different methods of prediction, the numerical solution offers promise. A numerical method is constructed for predicting the processes of interest. {NOT FOR LOAN FROM AIVC}
KEYWORDS heat transfer, numerical modelling

#NO 3855 Smoke spread simulation in a covered sports stadium.
AUTHOR Pericleous K A, Worthington D R E, Cox G
BIBINF UK, BHRA, Cranfield, Seminar and Workshop, 78 June 1988 held in London, 10pp, 8 figs, 2 tabs, 12 refs. #DATE 00:06:1988 in English
ABSTRACT The field modelling technique for predicting the temperature distribution and smoke movement in enclosures containing a fire source is validated against experiments carried out in a fully instrumented sports building covered by an air supported dome. The building is oval in plan and the dome has an ellipsoidal shape. A 2MW methanol pool fire located centrally on the floor of the building was used to obtain detailed measurements of temperature at a number of locations. The mathematical model simulates the transient problem in three dimensions using two different finite volume grids. The first grid is a polar cylindrical one with cells partially blocked to simulate features not coincident with grid lines. The second uses a nonorthogonal grid which follows closely the contours of the building. Results are abtaincd for prefire, fire and postfire conditions and the two grid solutions are compared with experiments. Qualitative agreement is good throughout and trends are correctly simulated. Quantitative agreement is also good in all areas except in common with earlier studies in the immediate vicinity of the fire source. The body fitted grid solution predicts correctly the lack of stratification due to strong convection along the ceiling.
KEYWORDS mathematical model, smoke, fire, sports building

#NO 3857 Calculations of the temperature and flow fleld in a room ventilated by a radial air distributor.
AUTHOR Reinartz A, Renz U
BIBINF UK, International Journal of Refrigeration, Vol 7, No 5, September 1984, pp308312, 8 figs, 13 refs. #DATE 00:09:1984 in English
ABSTRACT In ventilated or air conditioned rooms optimal conditions of temperature, humidity and air velocity are required. In the present study the behaviour of a jet emerging from a radial plate distributor and the resulting air flow in the room were investigated. To predict the behaviour of the air flow a numerical scheme was used to solve the conservation equations for mass, momentum and energy with the ke turbulence model. The numerical results are compared with available experimental data.
KEYWORDS calculation techniques, temperature, air flow

#NO 3512 The constant tracer flow technique.
AUTHOR Sandberg M, Stymne H
BIBINF Building and Environment, Vol 24, No 3, 1989, pp.209219, 6 figs, 8 tabs 6 refs. #DATE 00:00:1989 in English
ABSTRACT This paper presents a simplified theoretical analysis of the constant tracer flow technique and a quantitative estimate of the accuracy when the ventilation air flow rate is constant. A system with a stable pressure regulator was used to keep the tracer gas flow rate constant. A number of tests were carried out in an unoccupied indoor

test house with ventilation air flow rates known with an accuracy of 2%. The aim of the tests was to explore the effect of incomplete mixing. The tests were carried out both with and without artificial mixing. Simultaneous measurements were carried out with the constant concentration technique. Finally, the entries in the socalled Tmatrix were determined by moving the tracer gas source from room to room.
KEYWORDS air flow, ventilation rate, tracer gas

Numerical heat transfer.
AUTHOR Shih T M
BIBINF USA, Hemisphere, Series in Computational Methods in Mechanics and Thermal Sciences, 1984,563pp.

#NO 4000 A guide to air change efficiency.
AUTHOR Sutcliffe H
BIBINF UK, Coventry, **AIVC** Technical Note 28, February 1990, 32pp, 9 figs, 7 refs. #DATE 00:02:1990 in English
ABSTRACT The main objective of this report was to provide a concise introduction into the subject of air change efficiency. Existing literature in this subject area is extensive, but it tends to be very detailed and is difficult for a newcomer to understand. Different authors also use different symbols and/or different definitions for the same concepts, which tends to confuse the reader. Little has been produced covering the basic ideas and concepts behind some of the terms used. Therefore this report aims to show the origins of the concepts used, provide proofs of the basic formulae and suggests standard symbols and definitions. Sandberg and Skaret differentiate between the terms air change efficiency and ventilation efficiency. Air change efficiency is a measure of how effectively the air prcscnt in a room is replaced by fresh air from the ventilation system whereas ventilation efficiency is a measure of how quickly a contaminant is removed from the room. This report covers only air change efficiency and related concepts. It should also be noted that the theory descibed in this report assumes a mechanically ventilated, air tight room where all the air enters and lcavcs via designated inlet and exhaust ducts.
KEYWORDS air change rate, ventilation efficiency

#NO 3110 Application of a multigrid method to a buoyancyinduced flow problem.
AUTHOR Thompson C P, Leaf G K, Vanka S P
BIBINF USA, Argonne National Laboratory, 1988. #DATE 00:00:1988 in English
ABSTRACT The numerical prediction of buoyancyinduced flows provides special difficulties for standard numerical techniques associated with velocitybuoyancy coupling. We present a multigrid algorithm based upon a novel relaxation scheme that handles this coupling correctly. Numerical experiments have been performed that show that this approach is reasonably efficient and robust for a range of Tayleigh numbers and a variety of cycling strategies.
KEYWORDS numerical modelling

#NO 3863 Vectorized multigrid fluid flow calculations on a Cray XMP/48
AUTHOR Vanka S P, Misegadcs K P
BIBINF USA, International Journal for Numerical Methods in Fluids, Vol 7, 1987, **pp635648, 8 figs, 4** tabs, 17 refs. #DATE 00:00:1987 **in** English
ABSTRACT This paper discusses aspects of vectorizing a recently developed calculation procedure for multidimensional recirculating fluid flows, The solution algorithm uses a coupled GaussSeidel relaxation operator in conjunction with the multigrid technique. The vectorization is performed on a CRAY XMP 48 using a single processor. In this paper, the vectorization techniques used and the observed speedups are presented for a model problem of laminar flow in a twodimcnsional square cavity. Large scale calculations with up to one quarter of a million finite difference cells (512 x 512) have been made in 45s of CPU time.
KEYWORDS air flow, calculation techniques

#NO 3854 Stansted terminal building and early atrium studies.
AUTHOR Waters R A
BIBINF UK, J of Fire Prot.Engr., Vol 1,No.2, 1989, pp6376, 12 figs, 3 tabs, 19 refs. #DATE 00:00:1989 in English
ABSTRACT The paper demonstrates how field modelling techniques can be used to understand smoke control strategies during the design phase of major buildings. The use of animation techniques that combine smoke and people movement results, for a particular fire scenario, to produce a computer driven animated colour portrayal of an evacuation is discussed.
KEYWORDS atrium, smoke

#NO 2525 Computation of air movement and convective heat transfer within buildings.
AUTHOR Whittle G E
BIBINF Int J Amb Energy, Vol 7, No 3, July 1986, p151163, 9 figs, 36 refs. #DATE 00:07:1986 in English
ABSTRACT A review has been undertaken of the numerical computation of the air movement and convective heat transfer within buildings. The fundamental conservation equations together with a turbulence model are described, and a numerical procedure for solving the elliptic partial differential equations is outlined. The literature on the subject, spanning the past twelve years, is reviewed. Attention is briefly drawn to examples of the use of numerical air movement codes.
KEYWORDS calculation techniques, air movement, heat transfer, turbulence, model, review, computer, prediction

#NO 1914 Investigating natural air flows on the basis of similarity theory in covered spaces with floor heating.
Untersuchung von naturlichen luftstromungen in fussbodenbeheizten hallenbauten auf der grundlage der ahnlichkeitstheorie.
AUTHOR Ziemssen C

BIBINF Stadt und Gebaudetechnik, 1985, Vol37, No 7, p104106. 3 figs, 9 refs. #DATE 00:00:1985 in German

ABSTRACT Describes qualitative experimental investigation of the air flow in a scale model representing a typical, average hall. Smoke was used to display the air flows. A mathematical model was also developed. Determination of the turbulent air flow in the model *confirms* the suitability of the mathematical model for use in quantitative experiments, in particular for measuring the heat flux density.

KEYWORDS air flow, air movement, mathematical modelling

Appendix 1. Summary of Flow Equations,

Continuity Equation:

$$\frac{\delta \rho}{\delta t} + \frac{\delta}{\delta x}(\rho u) + \frac{\delta}{\delta y}(\rho v) + \frac{\delta}{\delta z}(\rho w) = 0$$

Momentum Equations:

X Direction:

$$\rho\frac{\delta u}{\delta t} + \rho u\frac{\delta u}{\delta x} + \rho v\frac{\delta u}{\delta y} + \rho w\frac{\delta u}{\delta z} = \frac{\delta}{\delta x}(\mu\frac{\delta u}{\delta x}) + \frac{\delta}{\delta y}(\mu\frac{\delta u}{\delta y}) + \frac{\delta}{\delta z}(\mu\frac{\delta u}{\delta z}) - \frac{\delta p}{\delta x}$$

Y Direction:

$$\rho\frac{\delta v}{\delta t} + \rho u\frac{\delta v}{\delta x} + \rho v\frac{\delta v}{\delta y} + \rho w\frac{\delta v}{\delta z} = \frac{\delta}{\delta x}(\mu\frac{\delta v}{\delta x}) + \frac{\delta}{\delta y}(\mu\frac{\delta v}{\delta y}) + \frac{\delta}{\delta z}(\mu\frac{\delta v}{\delta z}) - \frac{\delta p}{\delta y}$$

Z Direction:

$$\rho\frac{\delta w}{\delta t} + \rho u\frac{\delta w}{\delta x} + \rho v\frac{\delta w}{\delta y} + \rho w\frac{\delta w}{\delta z} = \frac{\delta}{\delta x}(\mu\frac{\delta w}{\delta x}) + \frac{\delta}{\delta y}(\mu\frac{\delta w}{\delta y}) + \frac{\delta}{\delta z}(\mu\frac{\delta w}{\delta z}) - \frac{\delta p}{\delta z} - \rho g$$

Thermal Transport Equation:

$$\rho c_p\frac{\delta T}{\delta t} + \rho c_p u\frac{\delta T}{\delta x} + \rho c_p v\frac{\delta T}{\delta y} + \rho c_p w\frac{\delta T}{\delta z} = \frac{\delta}{\delta x}\left(k\frac{\delta T}{\delta x}\right) + \frac{\delta}{\delta y}\left(k\frac{\delta T}{\delta y}\right) + \frac{\delta}{\delta z}\left(k\frac{\delta T}{\delta z}\right)$$

Equation of State for an Ideal Gas:

$$\beta = \frac{-(\delta\rho/\delta T)}{\rho} = constant$$

Turbulence Equations:

Turbulent Diffusivity:

$$\nu_t = \frac{C_\mu K^2}{\varepsilon}$$

Kinetic Energy of Turbulence:

$$\frac{DK}{Dt} = \frac{\delta}{\delta x}\left(\frac{\nu + \nu_t}{\sigma K}\right)\frac{\delta K}{\delta x} + \frac{\delta}{\delta y}\left(\frac{\nu + \nu_t}{\sigma K}\right)\frac{\delta K}{\delta y} + \frac{\delta}{\delta z}\left(\frac{\nu + \nu_t}{\sigma K}\right)\frac{\delta K}{\delta z} + G - \varepsilon$$

Dissipation Rate:

$$\frac{D\varepsilon}{Dt} = \frac{\delta}{\delta x}\left(\frac{\nu + \nu_t}{\sigma_\varepsilon}\right)\frac{\delta \varepsilon}{\delta x} + \frac{\delta}{\delta y}\left(\frac{\nu + \nu_t}{\sigma_\varepsilon}\right)\frac{\delta \varepsilon}{\delta y} + \frac{\delta}{\delta z}\left(\frac{\nu + \nu_t}{\sigma_\varepsilon}\right)\frac{\delta \varepsilon}{\delta z} + C_1\frac{\varepsilon G}{K} - C_2\frac{\varepsilon^2}{K}$$

where:

$$G = \nu_t\left[2\left(\frac{\delta U}{\delta x}\right)^2 + 2\left(\frac{\delta V}{\delta y}\right)^2 + 2\left(\frac{\delta W}{\delta z}\right)^2 + \left(\frac{\delta U}{\delta x} + \frac{\delta V}{\delta y} + \frac{\delta W}{\delta z}\right)\right]$$

$$u = U + u' ; v = V + v' ; w = W + w'$$

Nomenclature

C_μ, C_1, C_2 = empirical constants

K = kinetic energy of turbulence (J/kg)

T = absolute temperature (K)

U,V,W = time average of instantaneous velocity (m/s)

c_p = specific heat of fluid (J/kg.K)

k = thermal conductivity of fluid (W/m.K)

p = pressure of fluid (Pa)

t = time (s)

u,v,w = velocity components in x,y and z directions (m/s)

x,y,z = Cartesian coordinate directions

β = coefficient of thermal expansion (K^{-1})

ε = dissipation rate of kinetic energy (m^2/s^3)

μ = laminar viscosity (kg/m.s)

ν = kinematic viscosity (m^2/s)

ρ = density of fluid (kg/m^3)

ν_t = turbulent diffusivity (m^2/s)

Appendix 2 Published Research Algorithms

Code / Source	Comments
SIMPLE (Shih 1984)	2-Dimensional demonstartion algorithm Published FORTRAN listing for laminar steady state flow. Text gives full details and discretisation to produce a general 3-D transient flow model.
EXACT3 (Kurabuchi et al 1990)	3-dimensional transient CFD Code. Momentum, buoyancy and turbulent flow. FORTRAN listing. For 386/387 PC or larger system. Cartesian Coordinates.
WISH3D (TPD-TNO Netherlands)	3-dimensional transient CFD Code. Momentum, buoyancy and turbulent flow. Cartesian Coordinates. SIMPLE Method. UNIX based computing systems.
KAMELEON (SINTEF Norway)	3-dimensional transient CFD Code. Momentum, buoyancy and turbulent flow. Cartesian Coordinates. SIMPLE Method. FORTRAN Code. Workstation/Mainframe.

Appendix 3 Commercial Algorithms

Code / Source	Code / Source
All Codes: K - ε Turbulence 3 - Dimensional Steady State / Transient Momentum/Buoyancy/Pollutant Transport	
Methods based on the SIMPLE Algorithm:	
FLOVENT Flowmerics Ltd 12 - 50 Kingsgate Rd Kingston on Thames Surrey Great Britain Tel +44 81 547 2682	PHOENIX (CHAM Ltd) Bakery Hse 40 High Street Wimbledon Village LONDON SW19 5AU Tel +44 81 947 7651
FLOW-3D UK Atomic Energy Authority Computer Science Division Harwell Laboratory UKAEA Oxfordshire OX11 0RA	STAR-CD Olympic House 317 Latimer Road LONDON W10 6RA Great Britain Tel +44 81 969 9639
FLUENT Creare Inc PO Box 71 HANOVER NH 03755 USA	ASTEC UK Atomic Energy Authority Computer Science Division Harwell Laboratory UKAEA Oxfordshire OX11 0RA

Appendix 3 Commercial Algorithms

Code / Source	Code / Source
All Codes: K - ϵ Turbulence 3 - Dimensional Steady State / Transient Momentum/Buoyancy/Pollutant Transport	
Finite Element Methods:	
FIDEP Fluid Dynamics International 1600 Orpington Avenue EVANSTON Illinois 60201 USA Tel +1 312 491 0200	

Printed In England By Coventry Printers (3/91 46491)

THE AIR INFILTRATION AND VENTILATION CENTRE was inaugurated through the International Energy Agency and is funded by the following thirteen countries:

Belgium, Canada, Denmark, Germany, Finland, Italy, Netherlands, New Zealand, Norway, Sweden, Switzerland, United Kingdom and United States of America.

The Air Infiltration and Ventilation Centre provides technical support to those engaged in the study and prediction of air leakage and the consequential losses of energy in buildings. The aim is to promote the understanding of the complex air infiltration processes and to advance the effective application of energy saving measures in both the design of new buildings and the improvement of existing building stock.

Air Infiltration and Ventilation Centre

University of Warwick Science Park,
Barclays Venture Centre,
Sir William Lyons Road,
Coventry CV4 7EZ,
Great Britain.

Telephone: +44 (0) 203 692050
Fax: +44 (0) 203 416306

Operating Agent for International Energy Agency, The Oscar Faber Partnership, Upper Marlborough Road, St. Albans, UK