2. Natural Ventilation and Computational Fluid Dynamics

2.1 Preamble

This chapter explains the motivation for the research by considering the evolution of natural ventilation in temperate/cool climates. Having established the birth of naturally ventilated buildings in a modern context, details of how the airflow in such buildings can be modelled are given.

The chapter begins by outlining the need for ventilation. The use and development of ventilation systems through history, with an emphasis on natural ventilation strategies driven by buoyancy, is given in the next two sections. The following section presents the various methods available for modelling natural ventilation and outlines the advantages and disadvantages of each. A final section then reviews the use of computational fluid dynamics (CFD) for modelling airflows in buildings, paying particular attention to buoyancy-driven flows.

2.2 The Need for Ventilation

In today’s technological society, the average person spends more than 90% of their time indoors (Awbi (1991)). It is therefore imperative, for acceptable human performance and well-being, that comfortable environments are maintained. Human comfort involves many parameters which include adequate fresh air provision (and exhaust of stale air) and maintenance of satisfactory temperatures.

Air exchange is necessary to remove stale air and odours, dilute the products of combustion (and respiration), remove harmful chemicals, and combat excess heat. Necessary ventilation rates vary according to the usage of a space, and there is legislation which governs minimum fresh air requirements. Recommendations for
minimum fresh air supply rates are given in volume A of the CIBSE Guide (1986). For example, in cellular offices, average residences and most dance halls, the recommendation is for a minimum fresh air supply rate of 12 litres per second per person.

### 2.3 Ventilation - a Brief Historical Review

Ventilation can be brought about either by mechanical means whereby large volumes of air are moved around a building by fans, or by natural means where it is driven by wind effects on a building and by temperature differences between internal and external air.

Natural ventilation occurs in almost all dwellings whether it is consciously intended or not. The earliest form of ventilation is probably that occurring in cave dwellings whereby the effects of wind would cause ‘exchange flows’ at the cave entrance. This bi-directional movement of air would remove stale air and replace it with fresh air. Caves also offered a natural cooling effect due to the absorption of heat into the rock which was important during the day in very hot climates. Camp fires used for warmth during the evening and winter continued to provide ventilation in the absence of wind - the large upward buoyancy forces produced by the fire would drive air out of the cave along its soffit causing fresh air to enter and take its place. A similar phenomenon also occurs in houses with open fires where air is drawn into the room through cracks in the building envelope to replace that which is exhausted via the chimney. Of course this type of natural ventilation offers little or no control over the amount of fresh air entering the space or, more importantly, the temperature of this air. Consequently draughts can be a major problem in older houses.

Until the early 19th century very little (formal) consideration was given to the ventilation of buildings in Britain. Some of the earliest attempts to control ventilation in large buildings can be found in the House of Commons building in 1836 (see CIBSE, 1997) and the Octagon in Liverpool in 1867 (Banham 1984). These buildings used
fires to drive large volumes of air upwards through chimneys. This *thermal syphon effect* was able to ‘pull’ stale air out of the occupied spaces through openings placed in the ceiling, and into ducts that lead to the base of the chimney. Fresh air, which could be pre-warmed using heat from a steam boiler, was then drawn into the space through low-level openings.

The use of heat sources to drive airflow requires sufficient space for the hot air to move uninhibited. Sufficient space could not always be found, especially in complex structures comprising many rooms. Consequently, research into fan-forced ventilation, which would drive the same volumes of air through *smaller* ducts, began receiving attention. Initially powered by steam and later gas, these fans and their necessary plant were so heavy and large that the only suitable location for them was in basement areas. This raised questions over the ‘roundabout and unscientific’ way in which air was drawn downwards into the basement over long distances from spaces high up in the building. This problem was resolved by the electrification of the 1880s which meant smaller plant sizes could be achieved, and even enabled fans to be placed in the room that was being ventilated.

During the 19th Century, the use of fans to ventilate large buildings such as The Royal Victoria Hospital, Belfast (Banham 1984) increased. Also, the large buildings necessary to house the heavy industry of the 1890s made it harder to get fresh air into the centre of the space by natural means, so again, massive ductwork was needed.

Using a ducted air system with fans it was also possible to exercise far more control over not only the distribution of air around a building, but also its temperature. Gas powered boilers were used for winter heating and methods for summer cooling were

---

investigated. The latter initially involved placing ice in intake ducts. Later, the use of a refrigeration plant to cool coils was used. It soon became evident however, that this did not always bring about comfortable conditions - the higher levels of humidity that resulted were a particular problem. Fuelled by the intense desire to make buildings more healthy and comfortable (rather than more energy efficient), the concept of full air-conditioning was addressed.

The earliest fully air-conditioned building is believed to be the Milam Building in San Antonio, Texas, completed in 1928 (see Banham 1984). This comprised a large refrigeration plant in the basement with air-conditioning plant distributed at regular height intervals over the twenty-one storey building.

The availability of reliable electric lighting in the early 1900s with its lower heat output helped deeper floor plans to be realised, which could be ventilated and cooled successfully using the new air-conditioning systems. Full air-conditioning quickly became the preferred method of controlling the indoor environment in large, non-domestic buildings - it was relatively easy to control and its operation simple to understand. Tall, rectangular ‘sky-scrapers’ rapidly emerged (e.g. the United Nations building, New York, in 1970). The size of air-conditioning plant required in such buildings can amount to several entire floors.

In the UK, the economic boom of the 1980s saw the appearance of opulent air-conditioned buildings such as the Lloyds building and the Canary Wharf Tower, both in London. However, people had begun to complain that they were not comfortable in their artificial environments. Complaints included: nausea; eye, nose and throat irritations; headaches; tightness of the chest; and general fatigue. This lead to the term ‘sick building syndrome’ (Raw et al. (1992)) and even to certain buildings being referred to as ‘sick buildings’. Amongst other parameters, sick building syndrome is often attributed to poor air quality brought about by insufficient ventilation, or poor maintenance of ducted air systems.
Recently, employers have become increasingly aware of the high costs associated with running an air-conditioned building (see BRECSU 1998 for an example of costs) and the amount of physical space required to house the equipment.

Proposals by the Department of the Environment in 1993 to reduce energy usage in buildings by limiting the use of air conditioning in large, non-domestic buildings (CIBSE, March 1993), and concern over ozone depletion, have encouraged a significant transition to more energy efficient (non air-conditioned) buildings.

### 2.4 Ventilation - Some Modern Practice Examples

Now that the way has been signalled for energy efficient, naturally ventilated buildings, architects and engineers have began revisiting the natural ventilation strategies of over 100 years ago, and ‘reinventing’ them in a modern context. The quest to introduce more natural light into buildings also offers opportunities for natural ventilation. Atrium spaces, although not usually designed with energy efficiency in mind, offer ideal circumstances in which buoyancy-driven natural ventilation can take place (Figs. 2.1 and 2.2).
Figure 2.1 Natural ventilation by means of a central atrium space (arrows show intended direction of airflow driven by buoyancy).

Figure 2.2 The library at Anglia Polytechnic University incorporates atria to bring about stack-induced cross ventilation. (Photograph Tony Weller/Builder Group).
The recently published CIBSE Applications Manual: “Natural Ventilation in Non-Domestic Buildings” (1997) contains many examples of buildings that are naturally ventilated. Amongst them is the new Learning Resource Centre at Anglia Polytechnic University completed in September 1994, which comprises four storeys with two central atria. The space is cross ventilated, in winter by means of enclosed perimeter heaters which draw in outside air through trickle vents. The air then moves across the floor plate and is drawn into the atrium spaces by the induced ‘stack effect’ (Figs. 2.1 and 2.2). Stale air is then exhausted at the top of the atria. When no pre-warming is required, the stack effect in the atria is used to draw in outside air through open windows at the perimeter of the surrounding office spaces.

The Engineering School at De Montfort University in Leicester, designed by Short Ford and Associates (Fig. 2.3), is said to be “one of the most important projects of the 1990s” (CIBSE, October 1993) due to the way “it could influence a new breed of environmentally sensitive buildings”. The building utilises different forms of natural ventilation, for example, shallow plan electrical laboratories are cross ventilated, but due to the deep plan nature of the central section of the building, natural ventilation is by stack effect induced by large exhaust chimneys. The two auditoria are also ventilated in this way. In the auditorium spaces, fresh ambient air enters via a plenum below raked seats (after pre-warming if necessary). A layer of warm air then forms just below the ceiling caused by the internal heat gains produced by occupants, electrical equipment, and lighting. This drives a flow out of the exhaust stacks, drawing in more fresh air below the seats as it does so. At the design stage of this building, salt bath modelling was used to gain confidence in the proposed natural ventilation strategy (see §2.5.4 for further details).
Figure 2.3 The School of Engineering at De Montfort University, Leicester:
(a) south elevation showing air inlets to an auditorium, (b) exhaust stacks from the central concourse.
(Photographs: Peter Cook).

The Inland Revenue Headquarters building in Nottingham (Fig. 2.4) adopts a mixed mode ventilation strategy. Upward moving air in the stacks located on all four corners of each building draws air across the floor plates and into the stacks where it is exhausted at the top. Air passes into the building through occupant-controlled windows and perimeter inlets in the floor where the flow can be assisted by fans if necessary. Placed at the top of each circular stack is an umbrella-type ‘lid’ which can be raised or lowered to control the volume flow rate of air through the space.
Since modern naturally ventilated buildings use innovative ventilation strategies which have not yet been ‘tried and tested’, it has become important to establish modelling tools capable of predicting the airflows and temperature distributions in these buildings.

### 2.5 Modelling Natural Ventilation

It is more difficult to size a natural ventilation system than it is a mechanical system, because in the latter, the designer has accurate information regarding the volume flow rates produced by items of equipment (fans, diffusers, duct combinations). In contrast, there is much more uncertainty regarding, for example, the combination of opening sizes and buoyancy-producing heat loads required to drive a certain volume flow in a naturally ventilated regime. It is therefore important that some form of model, physical or theoretical, is used to optimise key parameters, such as opening sizes.
and their position, prior to construction. The following techniques can all be used to aid the design of naturally ventilated buildings: Empirical Air Tightness Method; Simplified Theoretical Models; Zonal Models; Salt Bath Modelling; Wind Tunnel Testing; and Computational Fluid Dynamics.

2.5.1 Empirical Air Tightness Method

This is a very approximate method for estimating the rate of air infiltration into a space. The method uses extensive measured data which has allowed average infiltration rates to be correlated with building air tightness (envelope leakage rate at an applied pressure difference of 50Pa). Based on estimates of air tightness, overall infiltration rates can then be calculated as a function of the surrounding terrain and prevailing meteorological conditions.

2.5.2 Simplified Theoretical Models

In these models a single equation is used to estimate the total airflow rate into a space, \( q_v \), where

\[
q_v = \text{ELA} \times s(h) \quad \text{ach}^{-1}. \tag{2-1}
\]

The effective leakage area, ELA, is a measure of the air tightness of a building at a (reference) pressure difference of 4Pa. This is extrapolated from measurements made at an applied pressure difference of 50Pa. \( s(h) \) comprises factors that account for the flow into or out of a space due to temperature differences and wind. Use of this model is exemplified by Sherman (1980).

2.5.3 Zonal Models

Zonal models enable a building to be divided into several discrete zones connected by flow paths. A zone may be used to represent a room or corridor while flow paths might represent open doors, windows, and cracks (infiltration).
A set of equations to represent the mass flow $q$, between adjacent zones and between the interior and exterior are then defined by considering the pressure differences $\Delta P$ between the two zones. These possess the following general form:

$$q = C(\Delta P)^n$$

(2-2)

where $C$ and $n$ are constants.

Pressure differences responsible for driving mass flows through openings on a building façade are brought about by the wind:

$$P_w = \frac{\rho C_{pw} \nu^2}{2},$$

(2-3)

where:
- $P_w$ = wind pressure on façade relative to the static pressure of the free wind;
- $\rho$ = density;
- $C_{pw}$ = pressure coefficient due to wind; and
- $\nu$ = wind speed,

and by temperature differences (stack effect):

$$P_s = -\rho_0 g \times 273.15 \times \Delta h \times \left(\frac{1}{T_{\text{ext}}} - \frac{1}{T_{\text{int}}}\right)$$

(2-4)

where:
- $P_s$ = pressure difference due to stack effect;
- $\rho_0$ = reference density of air;
- $g$ = acceleration due to gravity;
- $\Delta h$ = height difference between high level and low level openings;
- $T_{\text{ext}}$ = external air temperature; and
- $T_{\text{int}}$ = internal air temperature.
This yields a total driving pressure (pressure difference) between the interior and exterior domains of \( \Delta P = P_w + P_s \). This pressure difference drives a mass flow of \( q \) according to Equation (2.2). The flow problem is then solved by ensuring that the sum of all the mass fluxes between a zone and its surrounding zones equate to zero, i.e. that the total flow entering each zone is equal to that leaving. Values of pressure in each zone are then found.

Examples of multi-zone models can be found in Walton (1994) and Allard (1990). Such models can be integrated with dynamic thermal simulation programs such as ESP-r (ESRU (1996)) which can be used to predict the variation with time of internal temperatures and other building energy related parameters.

### 2.5.4 Salt Bath Modelling

This is an experimental technique whereby perspex models of buildings (typically at a scale between 1:20 and 1:100) are inverted and placed in a large tank of fresh water. Brine solutions are then injected into the model to represent sources of heat. Using an inverted camera and dye to colour the brine, the flow characteristics can be visualised. The justification for the use of brine and the various scaling techniques necessary to obtain quantitative full-scale data such as air change rates, are given in Chapter 3.

Salt bath modelling has been used to assist at the design stage of several naturally ventilated buildings. For example, the School of Engineering at De Montfort University, Leicester (Lane-Serff et al. (1991)), the Cable and Wireless building in Coventry (Edwards et al. (1994)) and the New Energy Efficient Office at the Building Research Establishment, Garston (BRE Compact Disc (1997)).

The salt bath technique has also been employed for validating analytical models based on plume theory which can be used to predict some of the main parameters determining buoyancy-driven flows.
2.5.5 Plume Theory

This analytical method is rarely used alone for predicting airflows in buildings and was therefore not included in the list immediately preceding section 2.5.1. However, it was an integral part of the validation process in this research and is therefore introduced here. Further details are given in Chapter 3. The method can be applied to natural ventilation of simple geometries containing clearly defined sources of buoyancy such as line or point sources. The heat sources produce rising plumes for which equations are now well established (Shmidt (1941), Morton et al. (1956), Baines and Turner (1969), and Baines et al. (1990)).

The method begins by postulating a flow pattern and then uses the plume equations for volume, momentum and buoyancy flux to establish macroscopic parameters such as positions of stratification interfaces, and temperature gradients across such interfaces. It is also possible to use the plume equations to find properties such as air change rates through the space. The most noteworthy work carried out so far is that of Linden et al. (1990) who used plume theory to determine the flow in a simple box undergoing mixing and displacement ventilation driven by temperature differences. The contents of this work relevant to this research are discussed in more detail in Chapter 3.

Since plume theory involves approximations and empiricism (see Chapter 3), all the theory derived by Linden et al. (1990) and later Cooper and Linden (1996) and Hunt and Linden (1997) is substantiated using salt bath experiments.

2.5.6 Wind Tunnel Testing

This technique enables models of buildings and their surroundings to be subjected to varying (and controlled) wind speeds. Pressure measurements taken over the building façade help to inform parameters of the design such as position of openings and likely air change rates. Flow visualisation is possible using smoke injection. Pressure coefficients calculated from the measurements can be used for providing
boundary conditions at openings in computer based models of internal flows such as computational fluid dynamics.

2.5.7 Computational Fluid Dynamics

This is a detailed modelling technique used primarily for calculating velocities, temperatures and pressures at numerous locations throughout a space to give a detailed picture of the complete flow field. This is achieved by dividing the space up into small cells to create a two or three dimensional mesh. For example, in a space measuring $2.5m \times 2.5m \times 2.5m$, the cells might range in size from $0.05m \times 0.05m \times 0.05m$ to $0.3m \times 0.3m \times 0.3m$. The conservation equations governing fluid flow, namely mass, momentum and energy transfer, are then solved in each cell using the values for velocity, pressure and enthalpy in the neighbouring cells as boundary conditions.

In general, any variable that satisfies a conservation equation (Eq. (2-5)) can be solved for in this way to yield its distribution throughout the space, so, for example, the water vapour content in humid air, and concentrations of brine in water, can be determined.

\[
\frac{\partial}{\partial t} (\rho \phi) + \frac{\partial}{\partial x_j} (\rho u_j \phi) = -\delta_{ij} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) + S_j \quad (2-5)
\]

where:

- $\phi$ = arbitrary variable;
- $t$ = time;
- $x_j$ = spatial coordinate vector;
- $\rho$ = density;
- $\delta_{ij} = \begin{cases} 1 & \text{if } i = j \\ 0 & \text{if } i \neq j \end{cases}$;
- $p$ = pressure;

\( S_j \) = Source of \( \phi \).
\[ \Gamma_\phi = \text{diffusion coefficient}; \text{ and} \]
\[ S_\phi = \text{source term}. \]

When all of the equations in each cell have been solved satisfactorily, the results are viewed using some form of *post-processor* - a graphical software module incorporated into most CFD packages. This enables, for example, velocities, pressures, temperatures, and brine concentrations to be represented using vectors and contours.

### 2.5.8 Advantages and Disadvantages of Each Modelling Technique

Table 2.1 summarises each of the modelling techniques (except plume theory) and outlines the advantages and disadvantages of each. It is seen that computational fluid dynamics, although probably one of the most expensive techniques, offers great potential for building designers. “...Used with care and with the exercise of sound engineering judgement...” (Jones and Whittle, 1992) it can help to resolve many of the practical problems associated with naturally ventilated buildings such as: avoidance of draughts; correct sizing of openings; and adequate ventilation.
<table>
<thead>
<tr>
<th>Technique</th>
<th>Description</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simplified Theoretical Models</td>
<td>Simple (one-equation) method for calculating approximate airflow rates in a single zone space.</td>
<td>Quick, easy and cheap.</td>
<td>Lacks accuracy, including flow direction. No information on airflow within the space.</td>
</tr>
<tr>
<td>Salt Bath Modelling</td>
<td>Experimental technique in which perspex models of buildings are submerged in water and brine injected to represent sources of heat.</td>
<td>Visually easy to appreciate. Qualitative flow pattern within a space can be realised and easily photographed. Possible to measure specific parameters.</td>
<td>Expensive to construct accurate models. Time consuming to make geometric changes. Not possible to measure variables at arbitrary positions without intrusive apparatus. Practical experience required. Few equipped laboratories. No heat exchange with structure.</td>
</tr>
<tr>
<td>Wind Tunnel Testing</td>
<td>Experimental technique in which models of buildings and their surroundings are subjected to varying (and controlled) wind speeds.</td>
<td>Ideally suited for modelling external airflows. Flow visualisation possible.</td>
<td>Expensive and time consuming. Not possible to model internal flows.</td>
</tr>
<tr>
<td>Computational Fluid Dynamics</td>
<td>Computational technique which primarily solves conservation equations of mass, momentum and energy to find velocities, temperatures and pressures throughout the whole domain.</td>
<td>Potentially very accurate. Able to predict spatial information giving values for variables throughout the entire flow domain. Easy to investigate changes in geometry and operating conditions. Additional physical models (e.g. radiation models and contaminant dispersion) can be added with relative ease.</td>
<td>Experience required to achieve optimum performance from the code. Very computationally intensive. Detailed simulations of entire buildings or long transient simulations are currently prohibitive. Some validation/development is still required, particularly in the area of turbulence modelling and buoyancy-driven flows.</td>
</tr>
</tbody>
</table>
2.6 Use of Computational Fluid Dynamics for Modelling Building Airflows

This section offers a brief historical review of the application of CFD to modelling airflows in buildings. It focuses mainly on the use of CFD for modelling buoyancy-driven flows, highlighting areas of weakness and thereby pointing towards the motivation for this research.

The birthplace of CFD techniques in the UK is widely accepted as being Imperial College, London, where Nielsen (1974) is reported to be the first researcher to model air movement and heat transfer in buildings using computer code developed by Gosman et al. (1969). These early simulations were based on the stream-function/vorticity formulation which significantly simplifies the governing flow equations, and is applicable only in two dimensions. The simulations modelled turbulence using the standard $k-\varepsilon$ model devised by Launder and Spalding (1974) in which the transport of turbulent kinetic energy, $k$, and its dissipation rate, $\varepsilon$, are modelled. Later Nielsen used a finite difference form (see §A.2.3) of the governing equations, which included additional buoyancy terms, to model “Buoyancy-Affected Flows in Ventilated Rooms” (Nielsen et al. (1979)).

Ideriah (1980), again using finite difference techniques and a buoyant form of the $k-\varepsilon$ turbulence model, simulated flow in a square cavity for various Reynolds and Archimedes numbers\(^1\). The work showed that when buoyancy effects are significant ($Ar > 0.04$), numerical instability becomes a problem and under-relaxation techniques (control of the solution process - see §D.7.2) are required to ensure convergence of

\[^1\] The Reynolds number provides an indication of the relative magnitudes of inertia and viscous forces and is given by $Re = \rho UL/\mu$, where $U$ and $L$ are characteristic velocity and length scales respectively.
the iterative solution strategy. Also, agreement with experimental data degraded as the flow became more buoyancy-driven rather than momentum-driven. Ideriah suggested that these discrepancies were due to the turbulence modelling technique and suggested possible improvements.

Markatos et al. (1982 and 1984) offer physical justification for not using additional terms in the $\varepsilon$ equation when modelling buoyant flows, and obtained satisfactory agreement with experimental data for natural convection flow in a square cavity.

Many authors (e.g. Ideriah (1980), Fraikin et al. (1982), and Thompson et al. (1985)) have reported difficulties in solving buoyancy-driven flows with strong stratification. Jones (1985) explains why these difficulties occur and postulates that tight under-relaxation is necessary to solve buoyancy-driven flows. He suggests that, alternatively, such flows can be solved using the transient form of the governing flow equations and allowing a steady state to emerge. Jones derives a time-scale, $t = 1/f$ (where $f$ is known as the Brundt-Vaisala frequency) over which buoyancy-driven flows should be resolved.

Holmes and Whittle (1987), aware of the rapid growth of CFD use for building design, warned about the potential pit-falls using the phrase “garbage in, garbage out”. To help guide users of CFD programs they compiled a list of recommended steps to ensure “accurate and good quality” results. These recommendations included imposition of suitable convergence criteria and the use of experimental data for validation, or when this was not available, work to ensure that further refinement of the computational grid used for representing the flow domain would not influence the results.

The Archimedes number provides an indication of the relative magnitudes of buoyancy and inertia forces and is given by $Ar = g\theta L/\rho U^2$, where $\theta$, $U$ and $L$ are characteristic temperature difference, velocity and length scales respectively.
With the appearance of modern naturally ventilated buildings, interest grew in the use of CFD to model these flows and the accuracy which could be obtained. Davidson (1989) used a finite difference CFD code developed by Davidson and Hedberg (1986) to model buoyancy-driven displacement ventilation in a simple space containing a single heat source. The medium used was water as this enabled easier flow visualisation in the experiments which were used to validate the program's predictions. The results showed good qualitative agreement in that the CFD model accurately predicted the occurrence of stratification. However, discrepancies existed in the height of the interface separating the warm water above from the cooler water below. Davidson concluded that more work was needed to investigate the feasibility of using numerical techniques to model buoyancy-driven flows.

Whittle (1990) used CFD to analyse winter and summer conditions in a large atrium space. He notes that CFD codes do not deal well with strongly buoyant flows and that often long run times are needed to obtain a solution.

McGuirk and Whittle (1991) realised the need for benchmarks. They proposed a 2D benchmark in which a cool jet issues into a warm space at ceiling level. As the buoyancy of the incoming jet was increased, a degradation of convergence was observed and tighter under-relaxation was needed. In the most buoyant cases, a time-dependent flow was observed.

In 1992 Jones and Whittle reviewed the current status and capabilities of CFD for building airflow prediction. In the summary of their paper, they outlined what was needed in a CFD code to successfully model airflows in buildings and what they thought were the major short-comings and limitations of most codes. One limitation noted was turbulence modelling, and the need for faster convergence of buoyancy dominated flows.

In 1992, there was an important step forward in the use of two equation ($k−\varepsilon$) turbulence models. Yakhot, Orszag et al. (1992) used Renormalisation Group (RNG)
theory taken from their earlier work (Yakhot and Orszag (1986)), coupled with a double expansion technique, to derive modified forms of the $k$ and $\varepsilon$ equations originally derived by Launder and Spalding (1974). A detailed mathematical description underlying the RNG $k-\varepsilon$ model is far too complex to be included here and is not necessary in understanding its use in this research. Briefly however, the technique enables small scale degrees of freedom, say at the atomic level, to be represented in terms of larger scale motions, thereby removing the task of solving the motion at the atomic level. This removal and re-representation of degrees of freedom is continued iteratively until the resulting set of equations obtained can be solved on relatively coarser grids such as those used in a CFD model. Since the constants in the $k$ and $\varepsilon$ equations are derived from mathematical theory, the empiricism of the standard $k-\varepsilon$ model (Launder and Spalding (1974)) is removed. The model possesses high and low Reynolds number forms, which have the potential to obviate the need for wall functions in the low Reynolds number regions. It is also claimed that the RNG model handles stratification and swirl effects better than the standard $k-\varepsilon$ model. Various CFD software vendors have incorporated the RNG form of the $k-\varepsilon$ model into their codes and have reported great improvements in prediction over the standard model when compared with experimental data for certain flow types (for example, see FLUENT Inc. (1993) where flow over a backward-facing step with heat transfer is considered). Versteeg and Malalasakera (1995) note the excitement that has been brought about by the potential of the RNG $k-\varepsilon$ model, but emphasise the need for its extensive validation.

Jacobsen (1993) used the standard $k-\varepsilon$ model of Launder and Spalding (1974) to model displacement flows. He notes that the short-comings of the standard model are inherent in its eddy viscosity concept because isotropic turbulence is assumed. This implies that turbulence acts equally in all directions, (showing no preference to the direction of motion or gravity force). He also suggests that some discrepancies are due specifically to an ill-defined $\varepsilon$ equation. Despite these short-comings he proposes that the standard $k-\varepsilon$ model is a good compromise between accuracy
and the risk of numerical instabilities which is a feature of the more CPU-intensive
techniques, especially turbulence models which consider the motion of individual
Reynolds stresses and heat fluxes (see, for example, Murakami et al. (1992)).

Over the last few years, developers of commercial CFD codes have improved the
modelling techniques used to represent turbulence. They are also continually
improving the user interface, thereby making CFD accessible to an ever wider user-
base.

2.7 Selecting CFD Software

It was not the aim of this research to develop and validate a new CFD code, rather, it
was to use a readily (commercially) available package, and to evaluate its ability to
model buoyancy-driven displacement ventilation. The selection process undertaken
is given in Appendix A, which identifies the important factors that were considered
along with the elimination process that took place. Although the search criteria will
change over time, and vary depending upon the intended CFD application, its
underlying methodology is potentially useful. Following the search, which took place
in March 1993, CFX, developed by Computational Fluid Dynamics Services
(Atomic Energy Authority, Harwell) was selected.

2.8 Summary

This chapter has stated the need for ventilation of buildings and briefly discussed the
progress of ventilation systems up to the present day. Government bodies, architects
and clients are becoming increasingly aware of the importance to reduce energy
usage and CO$_2$ emissions, resulting in an increasing number of naturally ventilated or
mixed-mode buildings being constructed. It is imperative, at the design stage of such
(innovative) projects, that architects and engineers can demonstrate to their clients
that adequate and appropriate ventilation of the occupied spaces will be achievable.
Computational Fluid Dynamics (CFD) is potentially a very useful tool for assisting in the design of naturally ventilated buildings. However, the published literature points to limitations in the technique. Although much work is being conducted to overcome these, such as the development of new turbulence models, much validation work is still required, both to test the new modelling techniques and to establish exactly where the limitations lie, and their severity. It is hoped that the research reported in this thesis will make a significant contribution to the evaluation of CFD techniques for modelling buoyancy-driven displacement ventilation.