NRPB-R292

Atmospheric Dispersion Modelling Liaison Committee

Annual Report 1995/96

NRPB-R292

National Radiological Protection Board

NRPB-R292

Atmospheric Dispersion Modelling Liaison Committee

Annual Report 1995/96

National Radiological Protection Board
Chilton
Didcot
Oxon OX11 0RQ

Approval date: November 1996
Publication date: May 1997

Stationery Office, £10.00
ISBN 0 85951 404 8

Preface

In 1977 a meeting of representatives of government departments, utilities and research organisations was held to discuss methods of calculation of atmospheric dispersion for radioactive releases. Those present agreed on the need for a review of recent developments in atmospheric dispersion modelling, and a Working Group was formed. Those present at the meeting formed an informal Steering Committee, that subsequently became the UK Atmospheric Dispersion Modelling Liaison Committee. That Committee operated for a number of years. Members of the Working Group worked voluntarily and produced a series of reports. A workshop on dispersion at low wind speeds was also held, but its proceedings were never published.

The Committee has recently been reorganised and has adopted terms of reference. The organisations represented on the Committee, and the terms of reference adopted, are given in this report. The organisations represented on the Committee pay a small annual subscription. The money thus raised is used to fund reviews on topics agreed by the Committee, and to support in part its secretariat, provided by NRPB. The new arrangements came into place for the start of the 1995/96 financial year. During its first year, the Committee placed contracts for three reviews. The technical specifications for these contracts are also given. The reports from its contractors are attached as annexes to this report.

The Committee intends to place further contracts in future years and would like to hear from those interested in tendering for such contracts. They should contact the Secretary:

Mr A Mayall
National Radiological Protection Board
Chilton
Didcot
Oxon OX11 0RQ

Contents

Annexes

Atmospheric Dispersion Modelling Liaison Committee

1 Organisations represented on the Committee

Amersham International plc
Atomic Weapons Establishment, Aldermaston
British Nuclear Fuels plc
Department of the Environment Northern Ireland
HM Inspectorate of Pollution
  (now part of the Environment Agency)
Health and Safety Executive
  Major Hazards Assessment Unit
  Nuclear Installations Inspectorate
Ministry of Agriculture, Fisheries and Food
Meteorological Office
National Nuclear Corporation
National Radiological Protection Board
Nuclear Electric
Royal Naval College, Greenwich
Rolls Royce and Associates plc
Scottish Nuclear
Scottish Office (HMIPI)
Urenco (Capenhurst)
Westlakes Research Institute

The Chairman and Secretary are provided by NRPB.

2 *Terms of reference*

1 To review current understanding of atmospheric dispersion and related phenomena and to identify suitable models for application primarily in authorisation or licensing, in the context of discharges to atmosphere resulting from nuclear industry activities.

2 The Committee shall consist of representatives of government departments, government agencies and primarily the nuclear industry. Each organisation represented on the Committee shall pay an annual membership fee of £1000.

3 The Committee will consider selected topics. These should be selected following discussion and provisional agreement at meetings of the Committee, followed by confirmation after the meeting. Where possible, it will produce reports describing suitable models for that topic. These will reflect either the views of an Expert Working Group appointed by the Committee or the outcome of a workshop organised on behalf of the Committee. The Working Group will determine who should be invited to speak at workshops, and to subsequently review their outcome and identify suitable models.

4 The money raised from membership fees and registration fees for the workshops will be used to support the Working Group, the drafting of reports, and any other matters which the Committee may decide.

3 Reports of the Working Group on Atmospheric Dispersion

Clarke, R H (1979). The first report of a Working Group on Atmospheric Dispersion: a model for short and medium range dispersion of radionuclides released to the atmosphere. Harwell, NRPB-R91 (London, SO).

Jones, J A (1981). The second report of a Working Group on Atmospheric Dispersion: a procedure to include deposition in the model for short and medium range dispersion of radionuclides. Chilton, NRPB-R122 (London, SO).

Jones, J A (1981). The third report of a Working Group on Atmospheric Dispersion: the estimation of long range dispersion and deposition of continuous releases of radionuclides to atmosphere. Chilton, NRPB-R123 (London, SO).

Jones, J A (1981). The fourth report of a Working Group on Atmospheric Dispersion: a model for long range atmospheric dispersion of radionuclides released over a short period. Chilton, NRPB-R124 (London, SO).

Jones, J A (1983). The fifth report of a Working Group on Atmospheric Dispersion: models to allow for the effects of coastal sites, plume rise and buildings on dispersion of radionuclides and guidance on the value of deposition velocity and washout coefficients. Chilton, NRPB-R157 (London, SO).

Jones, J A (1986). The sixth report of a Working Group on Atmospheric Dispersion: modelling wet deposition from a short release. Chilton, NRPB-R198 (London, SO).

Jones, J A (1986). The seventh report of a Working Group on Atmospheric Dispersion: the uncertainty in dispersion estimates obtained from the Working Group models. Chilton, NRPB-R199 (London, SO).

4 Specifications for technical annexes

A Atmospheric dispersion at low wind speed

A report is to be written for the Atmospheric Dispersion Modelling Liaison Committee covering atmospheric dispersion at low wind speed. The report is to take account of:

- views expressed at the workshop on this topic which was organised some years ago by the Committee,
- views expressed by two consultants appointed on behalf of the Committee to advise on the contents of the report.

The report should cover the following situations:

- atmospheric dispersion models for cases where there is a very small, but non-zero, wind, over the period of interest,
- atmospheric dispersion for cases where the vector average of the wind velocity over the period of interest is zero, but there are shorter periods where the wind speed is non-zero,
- the frequency of calm conditions in the UK, and anemometer response at low wind speeds.

B Application of computational fluid dynamics codes to near-field atmospheric dispersion

In order to widen the scope of safety assessments and emergency planning to include workers within the site boundary, there is an increasing need to undertake site specific studies of atmospheric dispersion within a few kilometres of the release point. In this region, the dispersion and deposition patterns will be strongly influenced by the layout of buildings and local topographical features. Such calculations could be undertaken in a variety of ways, for example using simple extensions to the

Gaussian plume model or other analytical techniques, computational fluid dynamics (CFD) models or wind tunnels.

Atmospheric dispersion models based on CFD codes may prove to be a suitable alternative to analytical models or to physical modelling. Several CFD codes are commercially available and may be acceptable for use in modelling atmospheric dispersion.

The work principally involves an examination of the adequacy of CFD to atmospheric dispersion in the near field, leading to a recommendation for ways forward in modelling such situations, with the following objectives:

- to review work reported in the open literature on the application of CFD codes to atmospheric dispersion,
- to carry out a critical appraisal of the work undertaken by an Unnamed Organisation on the adaptation and application of the PHOENICS codes to near-field atmospheric dispersion,
- to comment on the applications for which analytical models, CFD codes or physical modelling would be most appropriate,
- to make recommendations on the way forward and, in particular, to recommend priorities for future modelling improvements.

The work must consider dispersion and deposition from sources located within a group of buildings. Dispersion modelling both inside and outside the building complex must be considered.

C Rise of a buoyant plume from a building wake

The Working Group on Atmospheric Dispersion gave some guidance in its fifth report (NRPB-R157) on when a buoyant plume might rise out of a building wake. However, the Group could give no advice on how to calculate the concentration from the rising plume.

The object of the proposed work is to write a report identifying those aspects for which models could be recommended at present or with further work.

The report must include:

- a summary of the advice in NRPB-R157,
- a review of experimental and theoretical work since NRPB-R157 was written.

It must also identify:

- situations for which it is likely that a simple model could be recommended on the basis of existing knowledge,
- situations for which a model such that calculations can only be undertaken using a computer program could be recommended on the basis of existing knowledge,
- situations where there are reasonable prospects of recommending a model if further experimental work is carried out – in this case, an indication of the work required should also be provided,
- situations where it is unlikely that a model can be recommended, and for which physical modelling will be required.

ANNEX A

Atmospheric Dispersion at Low Wind Speed

J A Jones
NATIONAL RADIOLOGICAL PROTECTION BOARD

1 Introduction

The Atmospheric Dispersion Modelling Liaison Committee and its Working Group have recommended models for a number of situations. The models, described in a series of reports of which the first is NRPB-R91 (Clarke 1979), are all based on the Gaussian plume model of atmospheric dispersion. This model cannot be used when the wind speed is zero, as the basic equations include the reciprocal of the wind speed. However, the model is also inappropriate at low, but non-zero, wind speed as the assumptions on which it is based no longer adequately represent the physical processes involved. In particular, the model assumes that dispersion along the wind direction is small compared to advection by the mean wind. This assumption is not correct at low wind speed, particularly in unstable atmospheric conditions.

Dispersion behaviour in light wind conditions can be discussed from two different points of view, centred either on the difficulties of undertaking reasonable calculations or on the underlying science. This report is concerned almost entirely with the first of these points of view. It discusses what is meant by low wind speed, gives some information on the frequency of low wind speed conditions, and describes some dispersion models that can be used in such situations. This can be considered in two parts, namely the use of the 'standard' Gaussian plume model, with revised values for parameters where necessary, and other relatively simple models that can be used in some situations. The difficulties of modelling plume rise and the effects of buildings and non-uniform terrain are also considered.

One possible definition of low wind speeds is provided by the speed when normal meteorological instruments begin to perform inadequately (Smith 1992). The instruments normally used at Meteorological Office sites are designed to be robust in all sorts of extreme weather conditions, and to record accurately conditions that are important for weather forecasting or for public safety. This they do well, but at the expense of not recording well (or even not recording at all) in low wind speeds. In particular, cup anemometers do not start to record until the wind speed is greater than about 4–6 knots (about 2–3 m s^{-1}). Wind vanes have a starting speed of about 1–2 knots. An indication of the wind speed can be obtained at low speeds by the responses of the anemometer and wind vane over a period of a few minutes. Thus the Meteorological Office classes situations where there is no movement of the anemometer but some movement of the wind vane as a speed of 2 knots, and situations where there is occasional movement of the anemometer with some movement of the wind vane as 3 knots. Situations where neither the wind vane nor the anemometer shows any movement are classified as 'calms'. The wind speed in these cases is likely to be less than 1 knot. More sensitive types of anemometers are available, but they are not generally connected to networks where the results are available to other potential users of the measurements.

Most plume dispersion models are based on the flux balance between the concentration, wind speed and release rate, which can be expressed as

$$Q = \int_0^\infty \int_{-\infty}^\infty (UC + \langle U^1 c^1 \rangle) \, dy \, dz \qquad (1)$$

where Q is the release rate, U is the mean wind speed, C is the mean concentration, $\langle U^1 c^1 \rangle$ is the longitudinal turbulent flux, which represents the effects of variations from the mean in the wind speed and concentration, and y and z are horizontal and vertical distances, respectively, at right angles to the mean wind direction.

The second term is ignored in the Gaussian plume model as it is assumed to be much smaller than the first term. Plume models are not appropriate when this assumption breaks down.

Two situations must be considered in dispersion modelling. The first case, which occurs in unstable atmospheric conditions, is where the mean wind speed tends to zero but the fluctuations, driven by thermal motions, do not. In this case, while the wind velocity, averaged over some period, can be zero the air is not at rest. Convective thermals are continually formed in the surface layers, and vertical and horizontal turbulent motions are associated with these thermals, and there can be a substantial amount of vertical mixing. The second case, which occurs in stable conditions, is where both the mean and the fluctuating components of wind speed become very small. In extreme cases it is possible for the wind field near the ground to stagnate completely, even though there is some wind at higher levels. Material released into the stagnant air can then only be diluted through self-induced mixing, and even small emitted momentum or buoyancy fluxes could dominate the overall dispersion.

One of the features of low wind speeds is that the wind direction tends to be extremely variable. This is because there is little or no contribution to the flow from any general features and random turbulent motions may generate what little flow there is. In this context, it is important to distinguish between the wind speed and the wind velocity. Speed is a scalar quantity, and so considers only the rate at which air is moving, but not the direction of movement. Velocity is a vector quantity, and so considers both the speed and the direction of air movement. A sequence of periods where the wind speed is not zero but the wind direction is not constant could result in a zero wind velocity when averaged over a longer period.

Information on the frequencies of calm conditions can be obtained from Meteorological Office statistics, as for example summarised by Smith (1992). This information suggests that calm conditions might occur for up to typically about 3% of the time, and that wind speeds could be less than 4 knots between 10 and 20% of the time at different sites. It also shows that wind speeds less than 4 knots are more likely in summer than in winter. Lines and Deaves (1996) have reviewed dispersion at low wind speed for the Health and Safety Executive (HSE). Their report also comments on the available meteorological data and discusses the frequencies of low wind speeds at different sites and in different periods of years, showing the variation both between sites and between the different periods at the same sites. One set of statistics is given which shows that the frequency of calm conditions at different sites varies between about 1 and 16%, with a mean of about 5%. A second set, for a different period and a different group of sites, shows a similar overall range. However, comparing the frequencies for the same site in the two sets of data shows differences – for example, at one site the frequencies of calms in the two periods are 8.5 and 13.6%. This variation shows the importance of obtaining information on the particular site for which concentration calculations are being undertaken, rather than relying upon generic data. It also suggests that there may be difficulties in using frequencies from historic data in analyses of the concentrations of future discharges.

2 Dispersion models

This section gives comments on methods which could be used to calculate concentrations in low wind speed conditions. Concentrations must be calculated allowing for the wind speeds and stability conditions that occur while the material is carried to the point in question. As material cannot travel far from the release point in low wind speed conditions, the models are only appropriate for use near the release point. The section is divided into separate subsections which relate to calculating the concentration from short releases in periods where there is a small, but non-zero, wind and where the average wind velocity is zero. A further subsection comments on the calculation of the contribution of low wind speed conditions to the average concentration over an extended period.

2.1 Conditions where there is a non-zero average wind velocity

The Gaussian plume model, in the form adopted in NRPB-R91 (Clarke 1979), represents the air concentration at ground level, C, by equation 2.

$$C(x,y,z) = \frac{2Q}{2\pi U_{10}\sigma_y\sigma_z} \exp\left(-\frac{y^2}{2\sigma_y^2} \right) F(h,z,A) \tag{2}$$

where U_{10} is the ten metre wind speed, σ_y and σ_z are the standard deviations of the cross-wind and vertical concentration distributions, respectively, Q is the amount of material released, and

$$F(h,z,A) = \exp\left(-\frac{h^2}{2\sigma_y^2} \right) + \exp\left(-\frac{(2A - h)^2}{2\sigma_y^2} \right) + \exp\left(-\frac{(2A + h)^2}{2\sigma_y^2} \right) \tag{3}$$

This cannot be used at zero wind speed as it includes the reciprocal of the wind speed. Hanna (1990) has suggested that the Gaussian plume model can be applied if σ_y, the standard deviation of the distribution of material at right angles to the wind direction, is replaced by the product σ_θ x, where σ_θ is the standard deviation of the wind direction and x is the travel distance. He also suggested that at low wind speeds, calculations using a polar rather than a rectangular coordinate system are more appropriate. In other words, the Gaussian model should be used with σ_θ to give the distribution of concentration with direction away from the average wind direction along an arc at a particular distance from the release point.

Hanna reported measurements of plume widths, sampled over a period of one hour, from low level releases. He suggested that the measurements are broadly consistent with the assumption that the cross-wind plume spread in light wind stable conditions is given by

$$\sigma_\theta = A/U \tag{4}$$

where A is a constant with a value of about 0.5 m s^{-1}, when the angle θ is given in radians.

The denominator of the Gaussian plume model expression for concentration contains the product σ_y U. Replacing σ_y by σ_θ x and using equation 2 leads to

$$\sigma_y U = 0.5\ x \tag{5}$$

and so the concentration at a particular distance becomes independent of the wind speed at low wind speeds. Hanna also pointed out that the assumption that the horizontal distribution is Gaussian breaks down as σ_θ increases beyond about 60° as there is then material in all directions from the release point. However, Hanna suggested that his simple formula, used with a Gaussian model, will adequately predict the peak concentration on an arc.

Smith (1992) has also examined data for the variation of wind direction at low wind speeds. He concluded that the standard deviation of wind direction for a one hour period can be represented by

$$\sigma_\theta^2 = (5°)^2 + (60°/U)^2 \tag{6}$$

This expression can also be used in Gaussian plume models, and predicts that concentrations increase as the wind speed decreases.

This discussion suggests that the NRPB-R91 model could be used with some simple modifications to calculate the peak concentration at a specified distance. The product σ_θ x would need to be used in place of σ_y, and one of the expressions above adopted for σ_θ.

Hunt has also suggested a model for use in unstable conditions where there is a small, but non-zero, wind velocity. This was originally described in a note prepared in 1987 for the Working Group, which was not published, but it is included here as an appendix. The model considers the source to be a series of instantaneous releases and uses the Gaussian formula for the concentration distribution from an instantaneous release. The concentration from the continuing release is then obtained by integrating the concentration from a series of instantaneous releases. Hunt gives the general formula for the concentration from a point source and from a line source at right angles to the wind direction. Asymptotic solutions are included for concentrations at distances large compared to the release height and for conditions of strong turbulence. There are also formulae for the concentration at ground level along the plume centreline, and for the variation of concentration with height above the ground. The model predicts that the concentration is non-zero upwind of the source.

Some authors (for example, Apsley 1987 and Sharan *et al* 1996) have given solutions of the diffusion equation for low wind speed conditions, which assume that the relationships between the standard deviation of plume size and the diffusion coefficient used at higher wind speeds are still appropriate. Both Apsley and Sharan *et al* showed that the standard Gaussian model can be obtained from their models in certain limiting conditions. These models can be used to investigate properties such as the extent of plume spreading upwind of the source. Sharan *et al* considered the ratio between the Gaussian limit of their model and their exact solution, for a range of values of two dimensionless parameters in the model. This ratio lies within the range from 0.9 to 1.4 for reasonable values of the parameters. Sharan *et al* also compared their model with observed concentrations within 200 m of the release point. They showed that the ratio of predicted to observed concentrations lies in the range from 0.5 to 2 for the majority of the cases considered.

2.2 Conditions where the average wind velocity is zero

Smith (1989) has developed a model for use in unstable, convective conditions, where the wind velocity averaged over a short period is either zero or has some very small, but unknown, value. The model was presented for a release of neutrally buoyant material from a stack. Although the wind velocity averaged over some period is zero, at any instant there is likely to be an effective wind in a random (horizontal) direction. There will also be a turbulent vertical component of wind. The model aims to calculate the probability that material will be carried to the point of interest, by considering the probability distributions of horizontal and vertical wind speed. The model gives the concentration in terms of the probability distributions of the horizontal and vertical winds, and the intermittency, and so gives the probability distribution for the concentration at a point, required for analyses of the consequences of short releases and for toxic releases where the response may not increase linearly with the concentration. Smith presented a number of diagrams showing the probability distributions of concentration at a series of distances from a source and the variation of the distributions with source height and averaging time. He extended this model to the situation where there is a mean wind, but it is insufficiently large to be measured with any accuracy by the instruments on site.

Three figures, taken from his paper, are presented here as examples of the model results. The results are presented in terms of normalised quantities, rather than actual distances or concentrations. In all the results presented, distances are expressed as the ratio of the actual distance to the depth of the mixing layer, and the concentration is expressed in terms of $z_i^2 w_* c/Q$, where z_i is the depth of the mixing layer, w_* is the convective velocity, c is the actual concentration, and Q is the source strength.

Figure 1 shows the cumulative probability distribution of concentration at six distances from the source for zero averaging time, for a source whose height is one-tenth of the boundary layer height. The importance of source height is shown in Figure 2, which gives the probability distributions of concentration at one particular distance for three different release heights. Finally, the effect of averaging the concentration over different periods, from a continuous release, is shown in Figure 3, again for only one distance and one release height.

An indication of the average concentration from a release in unstable conditions could be obtained by considering that the dispersing material expands equally in all directions. The concentration then depends on the distance, but not the direction, from the source. The radius of the disk can be taken as $\sigma_u t$, where t is the travel time and σ_u, the standard deviation of the wind speed, is approximately equal to $0.5\, w_*$. The vertical dispersion could be described using the normal variation of σ_z, the standard deviation of the vertical distribution of material, with distance from the source.

2.3 Calculating the annual average concentration

A frequent use of the models recommended by the Working Group is to calculate the average concentration over an extended period, such as a year, using data on the joint frequency of occurrence of the different stability classes and wind speed bands. Such data generally include a category described as 'calm', for which no information on wind speed or direction is given.

Smith has developed a version of the model described in Section 2.2 which calculates the average concentration as a function of distance from the site. This is a version of the model described above which averages over the horizontal wind directions. It refers only to those calms which occur in unstable conditions.

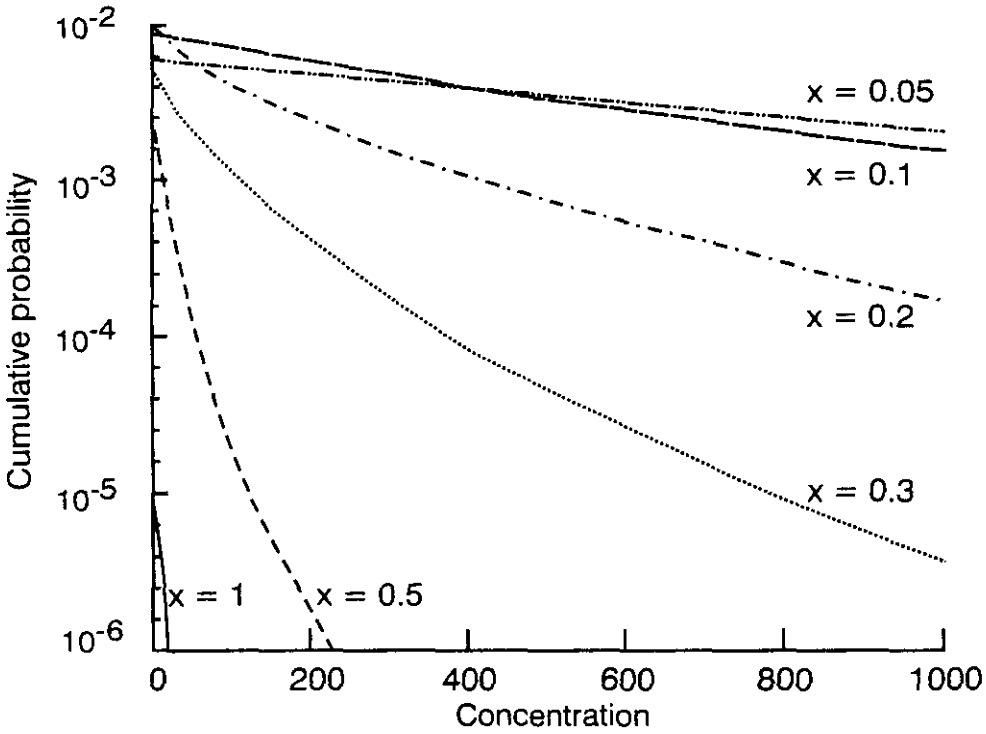


FIGURE 1 Cumulative probability distributions at six distances, x, from the source: source height = 0.1, averaging time = 0, with zero mean wind (distances and source height are normalised by the depth of the boundary layer)

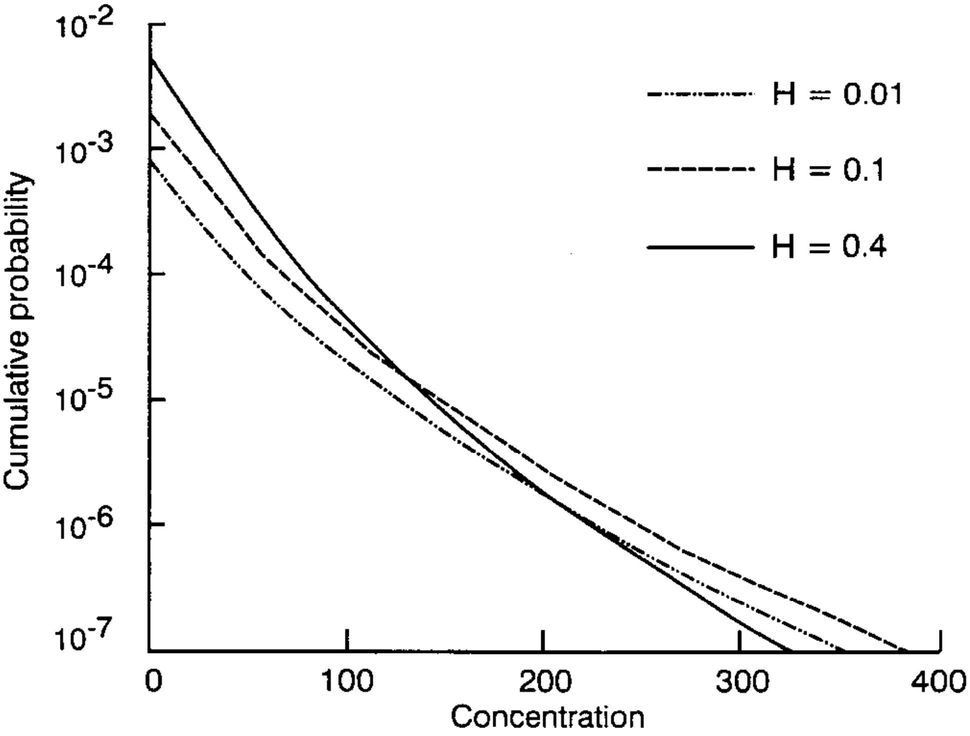


FIGURE 2 Effect of changing source height, H, on the cumulative probability distribution at distance x = 0.5, with zero mean wind (source heights and distance are normalised by the depth of the boundary layer)

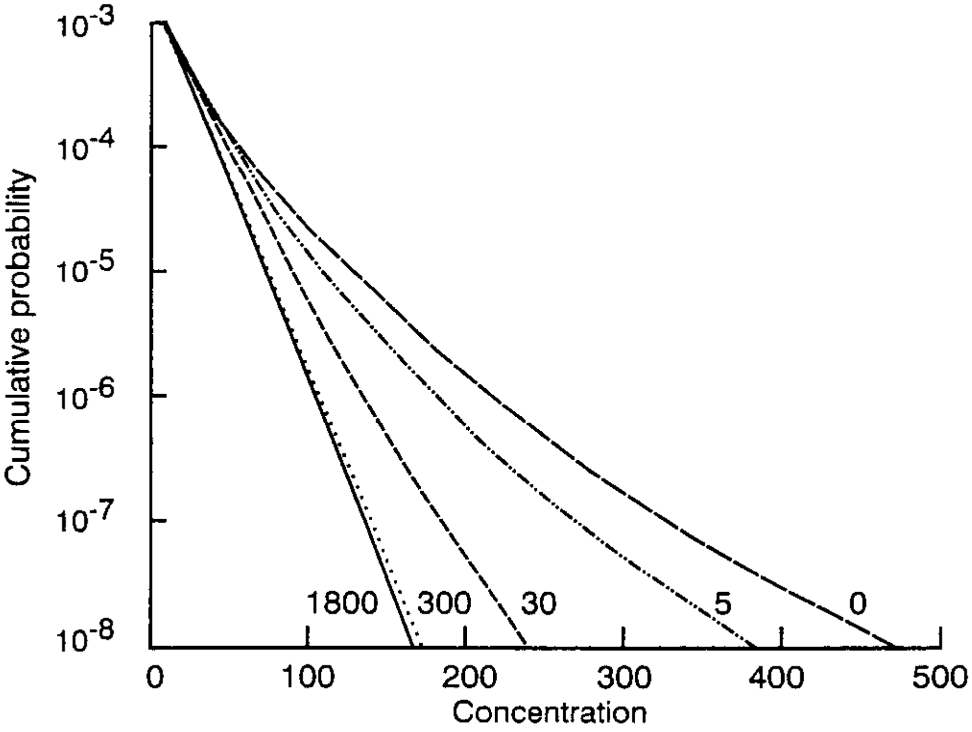


FIGURE 3 Effect of time-averaging (in seconds) on the cumulative probability distribution of concentration: source height = 0.1 and distance = 0.5 (distance and source height are normalised by the depth of the boundary layer)

Hunt's model could also be used for calculating the annual average concentration by averaging his formulae over direction from the site.

An alternative, and very simple, method of calculating concentrations is to add the periods of calm conditions into one of the higher wind speed bands. This is reasonable as the material cannot travel far away from the site during calm periods, and so must travel under the influence of the wind following the calm period. The travel is more likely to occur with a low wind speed than a high wind speed, and so the most reasonable method is to include the calm periods in the band representing the lowest non-zero wind speed. This method would underestimate the concentration at distances which could be reached by material travelling in conditions of effectively zero wind velocity, and so should only be used beyond such distances.

2.4 Summary of dispersion models

This section has reviewed the dispersion models available in the literature for dispersion in low wind speed conditions. Most of the work described relates to dispersion in unstable conditions. The models of Hunt and Smith are explicitly stated to be for use in unstable conditions. Only the work of Hanna explicitly considers low wind speed, stable conditions, and his work is limited to descriptions of the horizontal spread of the plume. The models based on the solution of the diffusion equation could apply in either stable or unstable conditions, but the validation described in the papers is restricted to unstable conditions.

A number of simple methods have been suggested for calculating the contributions of releases in 'calms' to annual average concentrations from continuous releases. The very simple nature of these methods must be stressed.

A major difficulty in any modelling of dispersion in low wind speed conditions is to determine the actual conditions at the time and position of the release. In particular, the models require estimates of the wind speed and turbulence levels. Lack of detailed information on the atmospheric conditions may be a major contributor to the difficulty in attempting to predict the air concentration in any particular situation.

3 Other effects

3.1 Plume rise

The Working Group has previously recommended models for plume rise (Jones 1983) based on work by Briggs (1984) and Moore (1980). The plume rise formula recommended includes the reciprocal of the wind speed, and so predicts an infinite rise at zero wind speed. Briggs considered what he termed 'vertical plumes' and 'bent-over plumes'. Vertical plumes refer to plumes rising in conditions of zero wind speed, while bent-over plumes refer to the normal situation of plumes rising in conditions with a non-zero wind. He considered the rate at which plumes rise near the source, together with a variety of situations which terminate the rise, giving equations for both vertical and bent-over plumes where necessary.

Briggs discussed several cases where the rise is limited by stable stratification, covering rise through a constant density gradient, through a density jump and the rise into an elevated stable layer. The models and equations given in these sections could be used for most aspects of the rise of a plume from a point source in low wind speed, stable conditions. The case where the rise is limited by convective turbulence was also considered. Although Briggs did not explicitly give formulae for the case of zero wind speed, the method described would be appropriate for calculating the rise of a plume in low wind speed, unstable conditions.

As stated in Section 1, the wind speed well above the ground is unlikely to be zero when the wind speed at ground level is zero. Therefore, although a plume may start to rise in a region with zero wind speed, it is likely that the rise will carry it into a region of non-zero wind speed. This can be seen by observing plumes which remain visible for some time, such as those from power station cooling towers. They are sometimes seen to rise vertically before reaching a layer with a non-zero wind speed when the rise terminates and the plume travels horizontally. Briggs did not consider this situation explicitly. He did, however, consider the case of a plume rising into an elevated stable layer and into irregular stability profiles. The methods described there could be adapted to the case of rise into elevated layers of non-zero wind speed. In any specific case, it would be important to know the temperature structure of the atmosphere above the release point, as the rise will terminate when the plume encounters an elevated inversion of sufficient strength, but could penetrate weak inversions.

Recent developments in plume rise modelling are the so-called integral plume rise models, in which the basic equations describing the trajectory of the rising plume, and the spread around that trajectory, are solved numerically. These models explicitly consider the vertical profiles of wind speed, temperature gradient and other conditions, and so naturally apply to situations where plumes can rise into or through regions in which the atmospheric conditions change. The problem, however, is to determine the profiles, as in near-calm conditions the similarity theory approach, appropriate at higher wind speeds, may not be applicable.

3.2 Non-uniform terrain

Non-uniform terrain can affect the airflow in a number of ways, with the major effects occurring in stable conditions. One such effect is the katabatic wind. This occurs when the temperature on the higher parts of a hill or valley is lower than that at the bottom of the hill or lower down the valley. In this situation, which often occurs at night, the density of the air at the top of the hill is greater than that of the air at the bottom of the hill, and so the air can sink under gravity from the top to the bottom of the hill. This airflow is known as a katabatic wind, and can be particularly strong in long valleys with hills on three sides of the valley. In some cases, the flow in the valley can decouple completely from the flow above the valley, and the wind direction within the valley is then not governed by the general flow patterns above the valley system. The slopes involved do not have to be large – katabatic flows have been recorded with mean slopes of only 1%.

The airflow near an isolated hill can be very different in stable conditions and in neutral conditions. In neutral conditions, air approaching a hill tends to flow over the hill, so that even material dispersing close to the ground is carried over the hill. The distance between the centreline of a plume and the ground is reduced as the plume passes over the hill, but the plume centreline does not touch the ground. The concentration at ground level on the hill is greater than that at ground level in the absence of the hill. In very stable conditions the whole of the airflow cannot rise over the hill. Material which is travelling at some height above the ground will rise and pass over the hill, but material travelling below a critical height is unable to do so, and passes around the hill. The plume centreline can be brought very close to the ground on the hill, and the concentration at the ground can equal that on the plume centreline after the same travel distance over flat terrain.

The flow around hills in stable conditions can be described in terms of the Froude number, a stability parameter related to the temperature gradient, wind speed and hill height. Large stability effects are seen when the Froude number is less than about one, and this value can be attained relatively easily.

3.3 Buildings

Airflow around buildings in low wind speed conditions has not been studied extensively. Some comments on flow round buildings can be made from the experience with stable flow around and over hills, and from the few studies which have been carried out.

Building effects are unlikely to be very important in strongly convective conditions, as the high ambient turbulence levels quickly dissipate special flow features generated in the vicinity of an obstacle. However, it is unlikely that the flow in extremely stable conditions is similar to that in neutral conditions, and building effects can persist for long distances. This can also be demonstrated by considering the virtual source model for building effects (see, for example, Smith 1989). Simple calculations show that, in unstable conditions, plume sizes a few hundred metres from typical buildings are similar to plume sizes at the same distance from a source in the absence of a building. However, in category F and G conditions, the plume size about 1 km from a point source is around 10 m, and so comparable to the likely initial plume spread behind a typical building. This suggests that building effects in these conditions could persist for distances of this size. Analysis suggests that the near-wake may grow in slightly stable conditions, compared to neutral conditions, due to reduced atmospheric turbulence. It will then contract as the stability increases.

Section 3.2 described the flow over and round an isolated hill in stable conditions. Similar effects can occur near buildings, but Froude numbers of around one or less are only possible in extreme conditions. No consistent and significant stability effects for sources below roof level are observed in field studies, although they are readily apparent for more elevated emissions. In an aerodynamic sense, buildings are much more severe than most terrain features: for example, they usually have sharp edges and vertical faces. As a consequence, there is generally a stagnation point on the upwind building face and plumes from close upstream sources may well impinge on the building. In such cases, the air concentration on the building face is typically that which would have occurred on the plume centreline at that fetch in the absence of the building.

Material can only leave a source if it has a non-zero exit velocity or is sufficiently buoyant to rise from the immediate source location. The plume rise induced by this initial momentum or buoyancy becomes more important as the wind speed reduces, and could eventually make an important contribution to the subsequent dispersion of the material. Low wind speed, stable conditions often occur at night in conditions with little cloud. In these situations the temperature outside the building is likely to be much lower than that inside. No building is perfectly insulated, and in conditions where the outside temperature is low the normal heat losses from a building could be sufficient to cause plumes emitted directly from the building face or roof to rise out of any wake.

Robins (1994) reviewed those field experiments which were carried out in low wind speed conditions. He concluded that the experimental results show that the concentration in the near-wake is consistent with that predicted assuming neutral conditions, and that source location and wind direction are more important than stability effects in all cases. Robins also reviewed the available data on concentrations in the main wake. Again there is very little evidence that stability has any marked impact on the concentrations, for releases from sources below roof level. However, such effects are apparent for higher emission points. These observations suggest that the normal models for dispersion from buildings can be used in low wind speed conditions for releases at and below roof level.

It should be noted that buildings and other large items such as trees are very effective at draining momentum out of the lowest layers of the air in rather light wind speed, stable conditions. It is not uncommon in these circumstances to have a virtual calm below the height of the buildings or trees, while the wind above can be quite significant. The two layers are often separated by a turbulent zone several metres thick. The flow within the roughness elements, although very light, can be detached from that above, and may carry plumes gradually away in a direction other than that of the prevailing wind, particularly on slopes.

3.4 Dry deposition

A number of papers describe the theory of dry deposition, and show how the deposition velocity can be calculated from a knowledge of the turbulence structure in the lowest layers of the atmosphere. Underwood (1987) reviewed the theory and commented on the likely variation of deposition velocity with wind speed in neutral conditions. He showed that the deposition velocity of particles which do not fall under gravity is likely to increase approximately proportionally to the wind speed. Although he did not explicitly consider the situation in low wind speed conditions, his analysis suggested that the deposition velocity would become very low as the wind speed approached zero. This is reasonable in stable conditions where the lack of vertical turbulence inhibits the flow of pollutant to the ground. However, in unstable conditions the situation is less clear since vertical turbulence may persist. Underwood also considered the variation of deposition velocity with stability category for the same wind speed at some height above the ground, concluding that the deposition velocity is unlikely to differ by more than about a factor of two across the whole stability range.

The arguments above suggest that the deposition velocity might tend to zero as the wind speed tends to zero. This does not, however, mean that the deposition would also tend to zero, as the concentration increases inversely with wind speed. The deposition rate could tend to a limiting value independent of the wind speed.

4 Summary and conclusions

This report has described particular features of the atmosphere which occur when the wind speed is low, and considered how these features affect dispersion. Models for use in calculating concentrations in some low wind speed conditions, particularly for releases of non-buoyant material from point sources, are given. There are no simple methods for calculating dispersion in stable conditions as the wind speed drops to very low values. Further work on the likely importance of such conditions should be carried out. Remarks are also given on other aspects of dispersion, such as releases from buildings or plume rise, in low wind speed conditions.

5 Acknowledgements

Dr A G Robins (EnFlo, University of Surrey) and Dr F B Smith acted as consultants in the preparation of this report. Their contributions are gratefully acknowledged.

6 References

Apsley, D D (1987). Diffusion in light wind conditions. Leatherhead, CEGB (National Power), Report TPRD/L/APM/0328/M87.

Briggs, G A (1984). Plume rise. IN Atmospheric Science and Power Production (D Randerson, Ed). Oak Ridge TN, Technical Information Center, US Department of Energy, DOE/TIC 27601.

Clarke, R H (1979). The first report of a Working Group on Atmospheric Dispersion: a model for short and medium range dispersion of radionuclides released to the atmosphere. Harwell, NRPB-R91 (London, SO).

Hanna, S R (1992). Lateral dispersion in light-wind stable conditions. *Il Nuovo Cimento,* **13,** C(6), 889–902.

Jones, J A (1983). The fifth report of a Working Group on Atmospheric Dispersion: models to allow for the effects of coastal sites, plume rise and buildings on dispersion of radionuclides and guidance on the value of deposition velocity and washout coefficients. Chilton, NRPB-R157 (London, SO).

Lines, I G, and Deaves, D M (1996). The implications of dispersion in low wind speed conditions for quantified risk analysis. London, HSE, CRR 8000/035.

Moore, D J (1980). Lectures on plume rise. IN *Atmospheric Planetary Boundary Layer Physics, Developments in Atmospheric Sciences*, Volume 11 (A Longhetto, Ed). Amsterdam, Elsevier.

Robins, A (1994). Flow and dispersion around buildings in light wind conditions. IN *Stably Stratified Flows: Flow and Dispersion over Topography* (I P Castro and N J Rockliff, Eds). Oxford, Clarendon Press, pp 325–58.

Sharan, M, Yadav, A K, Singh, M P, Agarwal, P, and Nigam, S (1996). A mathematical model for the dispersion of air pollutants in low wind conditions. *Atmos. Environ.*, **30,** No. 8, 1209–20.

Smith, F B (1989). Short range diffusion in convective light-wind conditions. IN *Air Pollution Modelling and its Application VII* (H Van Dop, Ed). New York, Plenum Press, NATO/CCMS, Volume 13.

Smith, F B (1992). Low wind-speed meteorology. *Met. Mag.*, **121,** 141–51.

Underwood, B Y (1987). Dry deposition to vegetated surfaces: parametric dependencies. Culcheth, UKAEA, SRD R 442.

APPENDIX

Atmospheric Diffusion from a Steady Source in a Turbulent Airflow at Low Mean Speeds

J C R Hunt
UNIVERSITY OF CAMBRIDGE AND THE METEOROLOGICAL OFFICE

1 Introduction

The current Gaussian plume models are based on the assumption that the mean wind speed is greater than the turbulence velocities. This is not a good assumption when there is strong thermal convection and a weak wind speed. For example, the rms vertical turbulent velocities at a height of 30 m might be about 0.5 m s^{-1}, so that if the mean speed is less than 0.5 m s^{-1}, the conditions for the existing model would not be satisfied.

A simple modification of the usual Gaussian plume model is developed in this note to account for such conditions. For simplicity it is assumed that the mean wind speed and turbulence are constant with height, but also that the observed values of σ_z and σ_y can be introduced into the model to account, in an approximate way, for these variations.

There are other low wind situations, where there is neither turbulence nor a mean wind. These situations usually occur in very stable atmospheric conditions, and the proposed model is not suitable for these conditions. Under these conditions, most of the dispersion results from the fluid motions caused by the ejection process, whether it is a chimney or an accidental explosive release, or by buoyancy forces in the ejected material.

2 Results for a line source perpendicular to the wind direction

If homogeneous Gaussian turbulence, a uniform wind, and a steady source at ground level emitting Q units per second per unit width are assumed, then

$$C(x,z) = \int_{-\infty}^{t} \frac{2Q \exp\left\{-[(x - U(t - t'))^2/2\sigma_x^2 + z^2/2\sigma_z^2]\right\} dt'}{2\pi\sigma_x\sigma_z} \qquad \text{(A1)}$$

where σ_x and σ_z are the turbulence-induced displacements in the x and z directions of a fluid element after travelling for a time (t - t′). This solution is effectively an addition of many elemental puffs released at time t′ at x = z = 0. Near the source it can be assumed that $\sigma_x = \sigma_u(t - t')$ and $\sigma_z = \sigma_w(t - t')$. For points far from the source, provided that the wind is not exactly zero, it can be

---

Note prepared for the Working Group on Atmospheric Dispersion in June 1987 assumed that σ_x and σ_z have their usual forms. Then, assuming $\sigma_x = \sigma_u(t - t')$ and $\sigma_z = \sigma_w(t - t')$, equation A1 gives

$$C = \frac{Q \exp\left[-\beta^2\left(1 - \dfrac{x^2/2\sigma_u^2}{x^2/2\sigma_u^2 + z^2/2\sigma_w^2}\right)\right]}{2\sqrt{\pi}\,\sigma_u\sigma_w\,(x^2/2\sigma_u^2 + z^2/2\sigma_w^2)^{1/2}}\left[1 + \operatorname{erf}\left(\frac{x\beta^2/U}{(x^2/2\sigma_u^2 + z^2/2\sigma_w^2)^{1/2}}\right)\right] \tag{A2}$$

where $\beta = U/(\sqrt{2}\,\sigma_u)$. For computational purposes $\operatorname{erf} x \approx x$ for $x < 1$ and $\operatorname{erf} x \approx \pm 1$ for $|x| > 1$.

(a) Thus when $x/z \gg 1$ for finite $U \neq 0$

$$C \approx \frac{Q \exp[-z^2/(2x^2\sigma_w^2/U^2)]}{\sqrt{2\pi}\,U\,\sigma_w\,x/U}\,(1 + \operatorname{erf}\beta) \tag{A3}$$

If we write $\sigma_z \approx \sigma_w\,x/U$, substituting in equation A3 gives

$$C = \frac{Q \exp(-z^2/2\sigma_z^2)}{\sqrt{2\pi}\,\sigma_z\,U}\,(1 + \operatorname{erf}\beta) \tag{A4}$$

When $\sigma_u/U < 1/2$, this agrees with the standard Gaussian formula to within 5%. Tabulated values of σ_z could be used.

(b) For very weak wind, where $U/\sigma_u \ll 1$

$$C \approx \frac{Q}{\sqrt{2\pi}\,(x^2\sigma_w^2 + z^2\sigma_u^2)^{1/2}} \tag{A5}$$

(c) The ground-level concentration is given by

$$C = \frac{Q}{\sqrt{2\pi}\,\sigma_w\,|x|}\,(1 \pm \operatorname{erf}\beta) \quad \text{for} \quad \begin{matrix} x > 0 \\ x < 0 \end{matrix} \tag{A6}$$

As with Gaussian formulae, this expression should only be applied outside the source itself which always has a finite diameter. The expression A6 shows that diffusion upwind extends over a similar distance to that downwind when the mean wind speed is very weak. When U increases the concentration is multiplied by a factor $[1 - \operatorname{erf}(U/\sqrt{2}\sigma_u)]/[1 + \operatorname{erf}(U/\sqrt{2}\sigma_u)]$. So if $U = 1$ m s^{-1} and $\sigma_u = 1$ m s^{-1}, the concentration upwind is on average about 20% of the value at the same distance downwind.

(d) The variation of concentration with height above the source is given by

$$C = \frac{Q \exp(-\beta^2)}{\sqrt{2\pi}\, \sigma_u z} \tag{A7}$$

3 Results for a point source

For a point source, the assumptions used in Section 2 now yield

$$C = \frac{2Q \exp\left[-\beta^2\left(1 - \frac{x^2/2\sigma_u^2}{(x^2/2\sigma_u^2 + y^2/2\sigma_v^2 + z^2/2\sigma_w^2)}\right)\right] G(p)}{(2\pi)^{3/2}\, \sigma_u\sigma_v\sigma_w\, (x^2/2\sigma_u^2 + y^2/2\sigma_v^2 + z^2/2\sigma_w^2)} \tag{A8}$$

where $G(p) = \frac{1}{2}\exp(-p^2) + \frac{\sqrt{\pi}}{2}\, p(1 + \operatorname{erf} p)$

and $p = \frac{x\beta^2/U}{(x^2/2\sigma_u^2 + y^2/2\sigma_v^2 + z^2/2\sigma_w^2)^{1/2}}$

(a) When $x/z,\ x/y \gg 1$, downwind of the source,

$$C \approx \frac{Q \exp\left[-\left(\frac{y^2}{2\sigma_y^2} + \frac{z^2}{2\sigma_z^2}\right)\right]\left[1 + \operatorname{erf}\beta + \frac{1}{\sqrt{\pi}\beta}\exp(-\beta^2)\right]}{2\pi\, \sigma_y\sigma_z\, U} \tag{A9}$$

so when $\sigma_u/U < 1/2$, again the error is less than 5%. *As in point (a) of Section 2, we have written* $\sigma_y \approx \sigma_v x/U$ and $\sigma_z \approx \sigma_w x/U$. Tabulated values of σ_y and σ_z could be used.

(b) For very weak wind where $U/\sigma_u \ll 1$, in the vicinity of the source,

$$C \approx \frac{Q}{(2\pi)^{3/2}\, \sigma_u\sigma_v\sigma_w\, (x^2/2\sigma_u^2 + y^2/2\sigma_v^2 + z^2/2\sigma_w^2)} \tag{A10}$$

(c) The ground-level concentration on the plume centreline is given by

$$C = \frac{Q\sqrt{2}\,\sigma_u\left[\frac{1}{2}\exp(-p^2) + \frac{\sqrt{\pi}}{2}\, p(1 + \operatorname{erf} p)\right]}{\pi^{3/2}\, \sigma_v\sigma_w\, x^2} \tag{A11}$$

Note: $p = (xU/2\sigma_u^2)/(x^2/2\sigma_u^2 + y^2/2\sigma_v^2 + z^2/2\sigma_w^2)^{1/2}$ so that at ground level on the plume centreline, we have $p = \beta$ sign x.

(d) The variation of concentration with height above the source is given by

$$C = \frac{Q \exp(-\beta^2)}{\sqrt{2}\, \pi^{3/2}\, \sigma_u \sigma_v\, z^2/\sigma_w} \tag{A12}$$

This is a more rapid decrease with height than for a line source.

4 Estimation of the turbulence

In strongly convective conditions, the components of turbulence are determined by the 'heat flux velocity' w_*, which is defined in terms of the surface heat flux H, the boundary-layer depth h, the gravitational acceleration g, the density ρ, and the absolute temperature T_o.

$$w_* = \left(\frac{g\, H\, h}{\rho\, C_p\, T_o} \right)^{1/3} \tag{A13}$$

Values of H are given in NRPB-R91. (*Note:* $C_p \approx 10^3$ J kg^{-1} K^{-1} and $\rho \approx 1.2$ kg m^{-3}.)

For the horizontal components (if the shear turbulence is included)

$$\begin{aligned} \sigma_u^2 &\approx 0.3 w_*^2 + 6.3 u_*^2 \\ \sigma_v^2 &\approx 0.3 w_*^2 + 4.0 u_*^2 \end{aligned} \tag{A14}$$

For the vertical components

$$\sigma_w^2 \approx 1.8 w_*^2 (z/h)^{2/3} + 1.7 u_*^2 \tag{A15}$$

I would recommend using a value of σ_w^2 at a height equal to about σ_z, so this might require some iteration to obtain a realistic value.

ANNEX B

Application of Computational Fluid Dynamics to Near-field Atmospheric Dispersion

R C Hall
WS ATKINS SAFETY & RELIABILITY

---

1 Introduction

1.1 *Background*

In modelling atmospheric dispersion for use in safety assessments and emergency planning, there is an increasing need to undertake site specific studies of atmospheric dispersion within a few kilometres of the release point. In this region, the dispersion and deposition patterns will be strongly influenced by the layout of buildings and local topographical features. Such calculations could be undertaken in a variety of ways, for example, using simple extensions to the Gaussian plume model or other analytical techniques, computational fluid dynamics (CFD) models or wind tunnels. There are now several commercially available CFD codes which might be suitable for dispersion modelling. This report describes an appraisal of the adequacy of CFD for modelling near-field atmospheric dispersion.

1.2 Objectives and scope of work

The objectives of the study were:

(a) to review work reported in the available literature on the application of CFD codes to near-field atmospheric dispersion,

(b) to carry out a critical appraisal of work undertaken by an Unnamed Organisation to adapt and apply the PHOENICS code to near-field atmospheric dispersion,

(c) to comment on the applications for which analytical models, CFD codes or physical modelling would be most appropriate,

(d) to make recommendations on the way forward and, in particular, to recommend priorities for future modelling improvements.

Sources within a group of buildings, and their dispersion and deposition both inside and outside the building complex, have been considered.

The study has examined the issues which are considered to determine whether CFD is an acceptable tool for modelling dispersion. These considerations are the capability of the method for representing realistic cases, the practicability of carrying out the work, the accuracy which should be achievable and the uncertainty of the results in practice due to the way in which the technique is used.

1.3 Terminology

It is recognised that some of the terminology used in CFD modelling is rather specialised and may not be familiar to everyone in the more broadly based atmospheric dispersion community. A glossary of the terms used in this report is therefore included in an appendix.

2 Review of literature

The dispersion of toxic and hazardous gases in the vicinity of buildings and in complex terrain is the subject of a current Health and Safety Executive (HSE) research programme. It was recognised that CFD modelling provided a potential means of improving the understanding of building and topographical effects on dispersion. HSE therefore initiated research to evaluate the capabilities of current commercial CFD codes for this application, the accuracy which could be readily achieved and the sensitivity of the results to numerical and physical modelling aspects. The first phase of this research involved a literature review and was reported by Lines *et al* (1994). The second phase involved CFD modelling of a number of relevant test cases and was reported by Hall (1996) and Gilham *et al* (1996). Another relevant research project in which WS Atkins has been closely involved is the Evaluation of

Modelling Uncertainty (EMU) project. This was funded by the European Commission and, at the time of writing, was due to be completed at the end of September 1996. A preliminary description was given by Hall *et al* (1996). These two projects have been used as the starting point for the present review of scientific literature, with additional recent published work being identified by undertaking a search at the Science Reference and Information Service, London.

2.1 HSE research: Phase I – literature review

The first phase of the HSE work reported by Lines *et al* (1994) involved a review of scientific literature relating to this subject area. Information was obtained both for CFD modelling and for full-scale and model-scale tests. A few of the most interesting papers and the overall conclusions from the review are summarised below.

Numerous papers were identified describing CFD modelling of wind flows around buildings. Most dealt solely with wind loading, ie the prediction of surface pressures, and included comparisons between CFD predictions and wind tunnel data for simple building shapes. The two-equation high Reynolds number k-ϵ turbulence model was generally used. This involves the solution of transport equations for the turbulent kinetic energy, k, and the dissipation rate, ϵ, in order to determine a locally varying eddy viscosity. The model assumes isotropic turbulence. Many papers were found to give consideration to turbulence modelling and/or appropriate boundary conditions for atmospheric simulations.

A few papers described comparisons with full-scale data. The Texas Tech building and the Silsoe Structures building are both full-scale buildings which have been extensively instrumented in order to provide realistic data for the validation of CFD and wind tunnel modelling. Selvam and Konduru (1993) predicted wind loads on the roof of the Texas Tech building for wind directions at 20° intervals over the full 360° range. The computed mean pressure coefficients at upwind corners of the roof were underpredicted by as much as 60%, and it was concluded that other turbulence models needed investigation. Richards and Hoxey (1992) found that their CFD model (k-ϵ turbulence model) failed to reproduce the 'delta wing' vortices near the gable ends of the Silsoe Structures building in oblique winds, but in general adequately reproduced the variations of surface pressure with direction.

A relatively small number of papers presented practical site applications. One of these was a paper by Laurence and Mattei (1993) which included some details of a study relating to a site in the city of Nantes. The finite element code N3S (k-ϵ turbulence model) was used to model an area of the city which included a domed church with five streets leading to the square situated in front of the church.

In contrast to wind loading applications, relatively few CFD studies of near-field dispersion were identified. Two significant studies were those of Benodekar *et al* (1985, 1987). The first of these involved the development of two- and three-dimensional CFD codes for predicting the flow and dispersion in the vicinity of buildings, taking account of release buoyancy and atmospheric stratification. The three-dimensional code employed the k-ϵ turbulence model and a second-order accurate differencing scheme modified to suppress spurious numerical 'wiggles'. An analytical Gaussian plume model was embedded within the code to represent the initial stages of plume development downwind from a point source. The wind tunnel experiments of Robins and Castro (1977a,b), involving dispersion in the vicinity of a cube, were studied. Benodekar *et al* focused on a number of cases, including a passive release from an area source and releases from stacks located at the centre of the roof, with the wind blowing normal to the front face of the cube. Some attention was given to modelling the anisotropic nature of the real wind environment, by modifying the turbulence model and 'tuning' it using the wind tunnel measurements of the turbulence properties around the cube. Some aspects of the predicted results compared poorly with the experimental results, such as the velocities over the roof at its upwind edge. Insufficient mesh refinement (only about 15,000 cells were used) and inadequacy of the turbulence model were thought to be responsible for this. Quite good agreement with the experimental results was shown for the concentration fields in the stack release cases. The second study re-examined the cube problem for the stack release cases using a finer mesh (about 60,000 cells). There was some improvement in certain aspects of the predicted flow behaviour, but it is significant that the predicted surface pressures near the roof leading edge and the concentrations in the wake appeared to agree less well with the experimental results. Scenarios with the wind blowing obliquely to the front face of the cube were also considered. The code was also applied to a more realistic reactor building shape. The wind tunnel measurements of Hatcher *et al* (1978) carried out for the Engineering Organic Cooled Reactor (EOCR) building in Idaho, USA, were considered for this. Three scenarios were studied: an area source at ground level, a point source release at roof level, and a stack release (above building height). In each case, the peak ground-level concentrations were overpredicted in the cavity region downwind of the building. The accuracy of the CFD predictions was reported to be within a factor of two to three of the wind tunnel measurements.

A more recent application, using a commercial code (FLUENT), was described by Moros *et al* (1992) involving predictions of fugitive hydrocarbon emissions from oil storage tanks. In such tanks, the roof floats on the oil, thus when the oil level is low the roof settles below the level of the rim of the tank wall, forming a cavity. The tanks in question were 20 m high and 25 m in diameter. Three different levels of the roof below the top of the tank rim were examined. It was demonstrated that the detailed flow patterns in the cavity and the height of the roof with respect to the rim of the tank were important parameters in estimating the emissions from open floating roof tanks.

2.1.1 Conclusions

The review identified a large number of relevant papers and reports, most involving the use of research-based CFD codes and most incorporating the k-ϵ turbulence model. It was generally reported that modelled results agreed qualitatively with experimental data, but that the turbulence model was the main cause of discrepancies. To improve the predictions, a number of alternative turbulence models and wall boundary conditions were tested, but there was little agreement over such improvements. Deficiencies in mesh resolution were acknowledged in a few cases but most authors appeared to consider their meshes to be sufficiently fine.

2.2 HSE research: Phase II – CFD modelling

The second phase of the HSE research, described by Hall (1996), involved the application of a commercially available CFD code, STAR-CD, to a range of test problems of varying complexity. The emphasis throughout was placed on the capabilities and performance of a commercially available CFD code, rather than on what could be achieved using academic and research codes, such as that employed by Benodekar *et al* (1985, 1987). Three validation test cases were studied: a continuous passive gas release in the vicinity of a cube (Robins and Castro 1977a,b), a continuous release of dense gas from a ground source (McQuaid 1976), and a large-scale instantaneous release of dense gas interacting with a semi-circular impermeable fence (McQuaid and Roebuck 1985). The cube test case in particular was used to examine the sensitivity of dispersion results to different modelling strategies. The key points which arose are discussed below.

2.2.1 Meshes and numerical aspects

The domain needs to be large enough so that the results in the region of interest are not affected by the positions of the boundaries. Following the examples of other researchers such as Benodekar *et al* (1987), the inlet and lateral boundaries were placed at about five building heights away from the edge of the building. The downwind (outflow) boundary was placed at sixteen building heights downwind of the rear face of the building. For the upper boundary, since relatively low level, non-buoyant releases were being considered, the boundary was placed at five building heights above the roof.

The most critical regions for modelling the local wind environment around a building are the sharp leading edges where the flow is most likely to separate. Fine mesh resolution is therefore needed in these areas to capture the flow details accurately. The nature of the wall boundary conditions needs to be considered too when choosing the mesh spacing next to walls. Fine mesh resolution is also needed in the vicinity of gas sources or 'instantaneous' gas clouds to capture the large gradients of velocity, turbulence and scalar quantities. Coarse meshes result in numerical diffusion and excessive plume spreading rates. For site safety studies, it is usually important to be able to determine gas concentrations and doses at distances from a couple of hundred metres up to one or two kilometres from the source. Attention must therefore also be given to mesh resolution in the wake region, even though the velocities and gradients in general are likely to be lower and the required mesh resolution consequently less refined than that required near buildings and sources.

In practice, it is unlikely that the optimum level of mesh refinement can be achieved everywhere. The best compromise will depend primarily on the nature and magnitude of the release. For example, a high momentum jet release may dominate the local wind effects around a building, thus requiring finer cells near the jet in order to capture the spreading rate and trajectory, with coarser meshing being acceptable at the building walls. In contrast, the trajectory and spreading rate of a small-scale passive discharge will depend largely on the local wind effects around the building.

Accurate numerical schemes are needed to avoid excessive numerical diffusion. Most CFD codes incorporate a selection of differencing schemes such as the first-order accurate 'upwind differencing' scheme and some higher-order schemes. The upwind differencing scheme is computationally cheaper than higher-order schemes but is less accurate and introduces false diffusion. The most accurate schemes may be 'unbounded' and can introduce non-physical under- or over-shoots, for example, negative concentrations near a source. Alternatively, most CFD codes have a 'blended' scheme which combines a higher-order accuracy scheme with the upwind differencing scheme, the blending factor being derived from local gradients. Such schemes are designed to be bounded and so do not introduce non-physical behaviour, but they are less accurate than the pure higher-order schemes.

2.2.2 Inlet and ground boundary conditions

Usually only the lower 200 m or less of the atmospheric boundary layer is of interest. In such circumstances, it is common practice for CFD modellers to treat the flow as part of the constant stress surface layer and specify turbulence profiles accordingly. The following expressions were therefore used for k and ϵ:

$$k = u_*^2/C_\mu^{1/2}$$

$$\epsilon = u_*^3/\kappa z$$

where u_* is the friction velocity, κ is the von Karman constant, and the constant C_μ describes the ratio of turbulent kinetic energy to the magnitude of the Reynolds stresses.

In some other applications, such as those involving releases of buoyant gases or when the boundary layer depth is relatively small, the height range of interest may well extend beyond the surface layer and it may be better to specify more realistic profiles.

An important modelling consideration is that equilibrium boundary layer conditions should be maintained across the mesh to a reasonable degree. Failure to do this might otherwise cause significant flow field changes quite independently of building and gas release effects. Such a situation would lead to a loss of accuracy and, perhaps, incorrect interpretation of results. For the cube case, the equilibrium nature of the prevailing flow conditions was checked by carrying out CFD runs for the meshes without the cube and then comparing the inlet and outlet profiles for the mean velocity and turbulent kinetic energy. In these tests it was demonstrated that the turbulence profiles showed a fair degree of sensitivity to the choice of roughness parameter, whereas the velocity profiles were insensitive. It is interesting to note that the dispersion results obtained with the k-ϵ turbulence model with C_μ values of 0.09 and 0.03 are quite similar (the standard value used in the k-ϵ model is 0.09, while 0.03 is more representative of real atmospheric flows). Although the levels of turbulent kinetic energy prevailing in the two scenarios were dissimilar (about 0.78 $m^2 s^{-2}$ compared with 1.37 $m^2 s^{-2}$, respectively), the eddy viscosities were similar, suggesting that, for the k-ϵ model, the precise nature of the vertical turbulence profile may not be critical for dispersion applications.

2.2.3 Turbulence and dispersion modelling

When predicting atmospheric dispersion around buildings, the main issues for turbulence modelling are the effect of the building on local velocity and turbulence regimes, the prevailing anisotropic effects of the atmospheric surface layer, and the effect of density stratification (due to molecular mass or temperature effects) on turbulence regimes. The first two aspects were important in the cube test case, while the last was important for the dense gas cases. For the cube case, three turbulence models were compared: the standard k-ϵ model, the renormalisation group theory (RNG) model and a Reynolds stress model (RSM).

Most of the cube simulations were carried out using the k-ϵ model, which is the most widely used in industry. Its shortcomings are well documented (see, for example, Boysan 1993). When used to model flows with significant normal straining rates (eg large $\partial u/\partial x$), spuriously high generation rates of turbulence energy are predicted. This excess of turbulence energy is then convected downwind and can seriously degrade the solution. In wind engineering, this translates into an overprediction of turbulence energy where the wind impinges on the upwind face of a building. This was demonstrated by the predictions.

The RNG model is reputed to perform well in separated flows: for example, in the simple backward facing step problem, the RNG prediction agrees quite closely with experimental results, whereas the k-ϵ model underpredicts the length of the recirculation zone by about 20%. In the cube test case, there was a significant improvement in flow field predictions, with flow separation being predicted at the leading edge of the roof using RNG, unlike the predictions of the k-ϵ model. There was little difference, however, between the concentration predictions.

The RSM model predictions of flow were also better than the k-ϵ results. The predicted flow separation at the leading edge was similar to that predicted in the RNG results, but the velocities in the reversed flow were stronger. The predicted concentration results were better than those of the standard k-ϵ and RNG turbulence models, but there were still significant differences compared with the experimental results. The RSM model overpredicted the peak ground-level concentration by 80%. This may have been due to the differencing schemes used (whilst a second-order differencing scheme was used for flow and turbulence, a less accurate blended differencing scheme was used for the concentration equation) and the approximate nature of the boundary conditions used for the inlet turbulence profiles.

2.2.4 Anisotropic diffusion

Turbulence and diffusion in the atmospheric surface layer are markedly anisotropic, being characterised by turbulent velocity components in the along-wind, cross-wind and vertical directions ($u' : v' : w'$), typically occurring in the ratio 1 : 0.68 : 0.45 (Cook 1985). The anisotropic behaviour results in increased lateral spreading of a plume and reduced vertical spreading near the ground. The k-ϵ and RNG turbulence models provide only an isotropic treatment and therefore tend to give similar spreading rates in the along-wind, cross-wind and vertical directions. The net effect, shown by some of the cube results, is primarily an underprediction of the rate of decay of concentration in the streamwise direction.

The RSM model represents these anisotropic effects, since transport equations are solved for the full set of turbulent stresses. The RSM results were better, but the computational effort was considerably greater due to the extra equations to be solved and the increased model size due to the extra variables.

Benodekar *et al* (1987) achieved better dispersion results for the cube case than those obtained in the HSE study with any of the turbulence models. One possible reason for this could be the use of an anisotropic dispersion model with anisotropy ratios (ie ratios of the turbulent stresses in the along-wind, cross-wind and vertical directions) 'tuned' according to the Castro and Robins experiments. The approach involved retaining the standard isotropic k-ϵ turbulence model, but modifying the transport equation for scalar concentration to include anisotropic eddy diffusivities. The eddy diffusivity in the vertical direction was obtained from the standard k-ϵ turbulence model. The ratios of the along-wind and cross-wind diffusivities to the vertical diffusivities were taken from the measured turbulent stresses in the longitudinal and lateral directions with respect to the main flow.

2.2.5 Conclusions

On the whole, good qualitative and reasonable quantitative agreement was achieved for the three validation test cases, except where atmospheric turbulence was the dominant mixing mechanism. For example, better results were obtained for the momentum-dominated phase of a large-scale dense gas release than for a small passive release. It was shown that the accuracy varied quite significantly with mesh resolution, inlet and ground boundary conditions, and turbulence modelling.

Predictions were also made for accidental releases at an actual industrial site. Both neutral and stable atmospheric conditions were simulated. Validation for realistic sites and non-neutral stability atmospheres was not addressed due to the absence of good experimental datasets.

Since the accuracy of predictions for realistic sites and scenarios could not be confirmed, it was concluded that it was not appropriate to apply CFD routinely within safety cases. However, it was acknowledged that CFD could be used to provide an insight into the detailed effects of buildings, release conditions and atmospheric conditions on gas dispersion. It was recommended that further research was needed on turbulence modelling for atmospheric dispersion applications. For example, the RSM model needs to be investigated rather more extensively, for stable as well as for neutral atmospheric conditions.

2.3 Project EMU

The EMU (Evaluation of Modelling Uncertainty) project is concerned with the uncertainties associated with CFD predictions for near-field atmospheric dispersion problems. The validity of such predictions is generally uncertain for two important reasons. Firstly, the way in which a CFD code is applied to a specific problem depends on the constraints on staff costs, timescales and computer resources, and can have a critical impact on the final results. Secondly, there is a disparity between the generally simple scope of model validation studies and the complexity of the actual industrial conditions.

The EMU project is funded by the European Commission, through its environment programme. A preliminary description was presented by Hall *et al* (1996). The main objectives of the project are to evaluate the spread in results due to the way in which a CFD code is applied, and to evaluate the accuracy of CFD predictions in large, complex gas dispersion situations. The general approach has involved:

- realistic industrial situations involving complex terrain and buildings,
- all four partners in the project using the same CFD code,
- realistic constraints on modellers,
- recording modelling decisions (to relate the differences in results to the different modelling strategies),
- wind tunnel modelling of selected test cases,
- quantitative evaluation of uncertainties.

CFD modelling was carried out in three stages of increasing complexity. The first stage involved a single L-shaped building on flat terrain with a neutral stability atmosphere and a 5 m s^{-1} wind speed (at 10 m above ground). A continuous release of neutrally-buoyant gas, a semi-continuous buoyant jet and an instantaneous large-scale release of dense gas were considered. In the second stage, the complexity of the site geometry was increased to include a second building, a cliff and a trench. A stably stratified atmosphere with a 2 m s^{-1} wind speed (at 10 m above ground) was also represented for some cases. The release scenarios included a continuous high momentum jet release of chlorine and an instantaneous release of 1 tonne of chlorine represented by a vapour cloud (28%) and an evaporating pool (72%). The final modelling stage involved an actual industrial site, which is located adjacent to cliffs at the coast, with low hills and a gully on the landward side of the site. The site itself slopes down towards the cliff and has numerous buildings. Two particular scenarios, both involving chlorine releases in a neutral atmosphere, were modelled: a steady 50 kg s^{-1} jet release and an instantaneous vapour cloud following the catastrophic failure of a 27 tonne storage tank.

Wind tunnel modelling has provided some useful data for the validation of CFD models. Of particular interest are the results for the industrial site. These include a passive release, a neutrally-buoyant jet release and a dense jet release (but CFD modelling has been carried out only for the dense jet case). The first of these corresponds to a virtually momentum-free release, while both the jet releases correspond to initial velocity ratios of about 15.

Regarding the CFD modelling strategies, it was interesting that during the first two stages, the different modellers tended to follow certain 'rules of thumb' for aspects such as the mesh distribution over the height of a building or at a source. It was apparent, however, that for the site application, the scale and complexity of the problem had generally forced the modellers to relax their 'rules' because they were simply no longer practicable.

2.3.1 Preliminary conclusions

Some preliminary conclusions of the EMU project have been published (Hall *et al* 1996, Cowan 1996, Cowan *et al* 1996). Firstly, the range of geometric scales of interest currently forces the user to compromise between accuracy and computational cost when designing the computational mesh. The resulting variation in numerical accuracy is the main cause of variation between the results obtained by different users. Computing power is therefore a key issue.

There is some evidence that the 'improvements' which one modeller might obtain, by modifying the turbulence model or details of the atmospheric boundary layer simulation, will lie within the typical spread between users. This suggests that the 'general' emphasis on the importance of physical modelling issues, such as the suitability of different turbulence models, may be rather misleading in the context of dispersion modelling.

The agreement between results of different modellers was better for a continuous release of dense gas in a neutral atmosphere, than when dealing with more difficult situations such as a transient gas release or a stably stratified atmosphere.

Reliable dispersion results require not only expertise in gas dispersion and CFD, but also familiarity with the use of the specific CFD program for the specific application of interest. Such familiarity can take a considerable time to achieve. The importance of quality assurance was stressed on several occasions.

The preliminary conclusions from the project are relevant for the present study, although they should be considered in the context of the general emphasis in the EMU project on large-scale dense releases. The scenarios of interest in the present study are not dominated by large-scale releases of dense gas; rather, low momentum releases of buoyant or neutrally-buoyant gas are also of interest. It is possible that the turbulence model would have an increased impact on the variability of results for such scenarios.

2.4 Open literature on CFD applications

A brief literature review was carried out to supplement the information gained from the studies described in Sections 2.1 and 2.2. A computer database search was carried out at the Science Reference and Information Service, London, concentrating on papers published in relevant journals and conference proceedings during the last three years. Selected relevant examples are described below under four headings: dispersion in the vicinity of buildings, urban pollution, topographical applications and turbulence modelling. The intention is to give an indication of current CFD activities in these areas, rather than a detailed account.

2.4.1 Dispersion in the vicinity of buildings

Götting *et al* (1995) used a CFD model to simulate the dispersion of a passive pollutant emitted from the stack of a power plant. The influence of the site buildings and the buoyant plume of an adjacent cooling tower were considered. The simulations were performed with the MIMO model, which employs the k-ϵ turbulence model and a finite volume structured mesh, adapted to follow the nature of the underlying terrain. The cooling tower has a height of 115 m and an outlet diameter of 69 m. The stack is 200 m tall. The highest building on the site is the 120 m high boiler house. Wind tunnel studies were carried out using a 1 : 1000 scale model under neutral stability conditions. The tracer SF6 was used to investigate the impact of the buildings and the interaction between the two plumes. Results from the CFD and wind tunnel testing were presented as longitudinal ground-level concentration profiles between 1 and 3 km downwind of the site and cross-wind profiles at 1.5 and 3 km. These CFD simulations are noteworthy in a number of respects: the large downwind extent of the domains used (ie greater than 3 km downwind of the source), the large size of the models ($120 \times 64 \times 48 = 368{,}640$ cells and $100 \times 96 \times 48 = 460{,}800$ cells) and the relatively short computing times (from 75 minutes with just the stack present to 4 hours for the most complex case with all the site buildings) achieved using a powerful vector computer. In comparison with the experimental results, the CFD model tended to overpredict concentrations downwind from the stack. The biggest discrepancies occurred when the plumes from the stack and the cooling tower dispersed side by side. In this situation, the stack plume becomes affected by the double vortex nature of the cooling tower plume, ie velocity vectors point upwards in the middle of a cross-section through the plume and downwards at both sides. The ground-level concentrations were overpredicted by CFD by about 30% at 2 km downwind from the stack and by about 50% at 3 km downwind. The paper does not give any details of the plume results above ground level.

Zhang *et al* (1996) compared CFD and physical modelling results for atmospheric flow and dispersion around a cube under stably stratified conditions. The CFD code, TEMPEST, used the standard k-ϵ turbulence model with a first-order accurate numerical scheme. The mesh was the same as that used for an earlier study under neutral stability conditions, reported by Zhang *et al* (1993). The physical modelling results corresponded to a uniform velocity profile upwind of the building, and were obtained from towing tank experiments. The Froude number was decreased from ∞ (neutral stability) to 3 (moderate stratification) and then further decreased to 1 (very strong stratification). The dispersion results compared well under weakly stratified conditions (Froude number ≥ 3), for a source located within the recirculation region behind the building, but poorly under strongly stratified conditions. It was argued that, because the Froude number will rarely be less than about 3 in the night-time stable boundary layer, stratification will rarely be a significant factor in influencing the flow structure in the near vicinity of a building.

The dispersion of dense gases at industrial sites was the focus of an earlier literature review outlined in Section 2.1. Following this review, other studies have been published in the open literature. For example, Perdikaris and Mayinger (1994) described comparisons between CFD and wind tunnel modelling of a continuous dense gas release from a short stack on top of a cuboid building. It was commented that 'agreement appears to be good'. Predictions were also shown for a time-dependent chlorine release at an industrial site. The effects of different wind directions and atmospheric stabilities were examined. It was concluded that the CFD method was flexible but that the computational effort was 'huge'. It was suggested therefore that CFD should be used for site planning rather than for emergency response predictions.

Fire and smoke modelling is another area of application for which CFD codes are increasingly being used, and numerous papers can be found in the open literature. Most are concerned with fires inside buildings, but a recent example of an external application was reported by Christolis *et al* (1995). They used the PHOENICS code to model the dispersion of pollutants around buildings on fire. A parametric approach was used to investigate the influence of wind speed on plume lift-off. The results which were presented included ground-level concentrations for distances of up to 800 m from the 30 m × 70 m × 8 m high building. No comparisons with experimental data were shown.

2.4.2 Urban pollution

There are several recent papers describing the use of CFD for modelling aspects of urban pollution. For example, Sini *et al* (1996) used the CHENSI code to study pollution in street canyons. The code, which uses the k-ϵ turbulence model, was previously validated for a series of thirteen reference cases including diffusive transport in circular and planar jets, plumes spreading in uniform and stratified atmospheric conditions, and recirculations in boundary layers over backward- and forward-facing steps, and two- and three-dimensional rectangular blocks. A two-dimensional approach was used to model the flow over and within a street canyon of 20 m depth and width varying between 6.6 m and about 300 m. Cell sizes ranged from 1 m × 1 m up to 7.5 m × 7.5 m. The domain extended 100 m upwind, 180 m downwind and up to 200 m above street level. The results were compared with wind tunnel data and good qualitative agreement was observed.

Eichhorn *et al* (1996) presented an example of dispersion from a point source at the centre of the open courtyard of a U-shaped building. The MISCAM code was used. The code employs the k-ϵ turbulence model, with a modification to the wall boundary conditions to achieve more realistic predictions of flow separation at building edges. This involves setting a zero longitudinal velocity at the upper front edges of buildings. Sedimentation and dry deposition can be taken into account. A comparison with some wind tunnel results was shown and 'almost exact agreement' was claimed. A traffic pollution case was also shown, but without any details of the CFD modelling.

Delaunay *et al* (1996) presented CFD predictions for traffic exhaust dispersion from road tunnels. They used the PHOENICS code with the 'dual time-scale' variant of the k-ϵ model (Chen and Kim 1987). This was intended to prevent the overprediction of turbulent kinetic energy in impinging flows. The mesh represented the region around the portal of a four lane road tunnel. A mesh resolution of 2–3 m was used in the horizontal directions and 1 m in the vertical direction. The resulting mesh had about 150,000 cells. Similar results were obtained using both first- and second-order accurate numerical schemes, suggesting that the resolution was adequate for the purpose. The results were found to be similar to those of the RNG model, but rather dissimilar to those of the standard k-ϵ model. Similar applications were presented by Jaeschke *et al* (1996) and Jicha *et al* (1996).

2.4.3 Topographical applications

A comparison between CFD and wind tunnel results for wind flows over and through model forests on two-dimensional hills was reported by Kobayashi *et al* (1994). The k-ϵ turbulence model was used, with non-standard constants. The effects of vegetation on airflow and turbulence were represented by inclusion of an additional drag force $C_D\alpha u|u|$ in the momentum equations (where C_D is a drag coefficient, α is the plant area density and u is the local velocity vector), together with additional terms in the k-ϵ turbulence model. A second-order accurate numerical scheme was used. The wind tunnel experiments used a 0.2 m high hill with 0.05 m high trees and the length of the hill was varied. Although the Reynolds number based on the hill height was only 8000 and therefore not comparable to that in the full-scale environment, it was judged that the flow was fully turbulent, so that the turbulence model performance could be studied. The suitability of the k-ϵ model for this application was discussed in some detail. The pronounced flow curvature over the hill and the overprediction of turbulent kinetic energy near the crest on the upwind slope where the wind tends to impinge on the hill were both perceived to be problems for the model. The lack of a universal, physically sound second-order turbulence model suitable for canopy flows was noted, together with the significant uncertainty in the description of forest properties, such as C_D values for trees. It was concluded that the model tested in the study was 'globally satisfactory'.

Tinarelli *et al* (1994) reported the development of a three-dimensional Lagrangian particle model, and presented a comparison with the US Environmental Protection Agency wind tunnel results for tracer dispersion in the lee of a two-dimensional hill.

A database of test cases for CFD validation is maintained by the University of Surrey on behalf of ERCOFTAC (European Research Community for Flow, Transport and Combustion). This includes some test cases for wind flow over hills, namely those of Thompson and Lawson (1990) and Almeida *et al* (1993). These have been used by some commercial CFD vendors to demonstrate the capabilities of their codes.

It is interesting to note that all the above applications involve smooth, symmetrical hills. There do not appear to have been any validation cases for more realistic, complex terrain.

2.4.4 Turbulence modelling

There is continuing basic research into flow and turbulence conditions in dispersion-related scenarios. For example, a recent paper about plumes in crossflows was presented by Savory *et al* (1996). This included a comparison between experimental and CFD results obtained with a Reynolds stress transport model and the k-ϵ model. The model used a second-order accurate numerical scheme and a scheme to prevent non-physical solutions in regions with rapidly changing conditions. The k-ϵ model was criticised for failing to predict the diffusion of vorticity in the longitudinal direction. Another point stressed was the importance of providing sufficient cells in the nozzle exit region in order to capture the local conditions reasonably accurately.

This paper is perhaps typical of a large number of theoretical papers produced in recent years, in which the standard k-ϵ model has been compared with alternative turbulence models and has been found inadequate. The published research can be divided broadly into two types. Firstly, efforts have been made to improve the k-ϵ model in some way, recognising that it is the most widely used model in industry. The second type of research has involved the development of more sophisticated techniques such as Reynolds stress modelling

Examples of improved k-ϵ models include the use by Cowan *et al* (1996) of the preferential dissipation modification (PDM) (Leschziner and Rodi 1981), the dual time-scale model (Chen and Kim 1987), the non-linear k-ϵ model (Basara and Younis 1992), and the RNG model (Boysan 1993). There seems to be no consensus on what improved k-ϵ model should be used. Another aspect which has been examined is the tuning of the constants in the k-ϵ model for a particular application. For example, Richards and Hoxey (1993) considered the constant C_μ which is related to the turbulent kinetic energy, k, in the atmospheric surface layer and the friction velocity, u_*, by the expression, $k = u_*^2/C_\mu^{1/2}$. For the real atmosphere, appropriate values of k/u_*^2 range from about 5.5 to 9. In comparison, the standard value of C_μ of 0.09 leads to $k/u_*^2 = 3.33$, while the value for C_μ of about 0.03 suggested by Richards and Hoxey leads to $k/u_*^2 = 5.8$. It is recognised that other constants should also be tuned for atmospheric flows (see, for example, Sutton *et al* 1986).

The alternative approach to extending the k-ϵ model has been to develop more sophisticated techniques, such as algebraic stress modelling (ASM), Reynolds stress modelling (RSM) and large eddy simulation (LES). Relatively few of the reported studies, however, are as directly relevant to dispersion applications as the study by Savory *et al* (1996) mentioned above and the earlier study by Goddard *et al* (1992). One of the more relevant benchmark problems is the flow past a surface-mounted cubical obstacle in a plane channel. Comparisons between k-ϵ, ASM and LES models for such a case were reported by Murakami *et al* (1992) and Murakami (1993). It was indicated that ASM or RSM were effective when the flow field was highly anisotropic and for predicting surface pressures or turbulence statistics, while LES provided greater accuracy at the expense of much greater computing times. More recently, Breuer *et al* (1995) described the comparison of k-ϵ (standard model and other variants) and LES results for a case involving flow past a surface-mounted cube, for which detailed experimental data (Re = 40,000) are available. The k-ϵ model calculations were carried out on a grid comprising 110 × 32 × 66 = 232,320 cells, while the LES calculations used a finer mesh comprising 165 × 65 × 97 = 1,040,325 cells. The smallest cell in the mesh used for the k-ϵ model simulations had a size of $(0.01h)^3$, where h is the height of the cube, compared to $(0.0125h)^3$ for the

LES mesh. It was reported that the LES computing times were about 200 times greater than those for the k-ϵ model for the mean quantities, and about 400 times greater for some of the turbulence statistics. As a consequence, LES simulations can still only be contemplated on powerful vector or parallel computers, whereas most other simulations (k-ϵ, RSM, etc) can be performed on workstations.

Considering commercial CFD codes, Freitas (1995) described a series of benchmark simulations which were completed using a variety of commercial codes by the vendors themselves. Two of the benchmarks involved turbulent flow problems. The paper highlighted the issues of mesh dependency and the accuracy of numerical schemes, but also illustrated an example of marked differences in details of flow field predictions obtained with the standard k-ϵ model, the RNG model and the RSM model. It was noted that, historically, it has been assumed that a series of simulations would at least identify the correct trend in spite of the limitations of the turbulence model. The differences between the results shown, however, prompted a warning that if insufficient attention was given to numerical accuracy and the use of an appropriate turbulence model, the trends predicted by one model might be different from the trends predicted with other models.

2.4.5 General comments

CFD modelling has been 'successfully' applied to large, complex industrial and urban sites, to topography and vegetation, neutral and stable atmospheres, and a wide variety of release conditions. The size of models has grown dramatically, up to over 1,000,000 cells in one study, reflecting a growing confidence in the usefulness of CFD as a dispersion tool. In contrast to the EMU project, little attention has been paid to the possibility of different users obtaining different results. Instead, most papers have focused on the turbulence model as representing the main area of difficulty.

3 Review of feasibility study

This section reviews a study undertaken by an Unnamed Organisation (1995), into the feasibility of using CFD for modelling near-field atmospheric dispersion. The study dealt with a site comprising numerous buildings spread over an area covering about 600 m × 600 m and predictions for five accident scenarios were presented.

It should be noted that the study was intended to demonstrate that CFD is able to model near-field dispersion adequately. For this reason, a coarse resolution model was considered to be acceptable. However, for the present review, the purpose of the study has not been taken into account, and some critical comments have been made. No criticism of the quality of the work is intended; rather, the study has been used as a convenient basis for a discussion of some of the key issues affecting CFD modelling of near-field dispersion.

3.1 General approach and problem specification

The CFD modelling was carried out using the PHOENICS code. Details of how to operate the software are described, including examples of the various input and output datafiles, plotting and run command scripts, and the user-specified subroutines for implementation of application-specific processes such as aerosol deposition and dose uptake calculations. It is interesting that this application makes quite extensive use of user-specified scripts and programming. The current trend is towards the use of a 'graphical user interface' (GUI), but while this can greatly ease and speed up some aspects of the model preparation process, it can make it difficult to check input data. Quality assurance is straightforward and therefore probably more reliable when using text files.

The computational approach for simulating dispersion involved two stages. Firstly, the wind field was solved for a number of different wind directions, in turn, and each of these solutions was stored in a library. In the second stage, the transient dispersion was simulated using the existing wind field solutions 'linked' together according to a specified time-varying wind pattern. The 'decoupling' of the wind and dispersion calculations assumes that the release has no effect on the wind field. As pointed out in the feasibility study report, this will not be valid for releases with significant source momentum or buoyancy effects. This will be very important in relation to the computational costs of simulating realistic scenarios.

3.2 Domain and mesh design

The main region of interest at the site measured 580 m × 570 m. The site was on two levels, the upper level being about 15 m above the lower level. Buildings of up to 30 m in height were located on the upper site level. This site region was placed at the centre of a mesh which extended 1000 m further away in each direction. Overall, the domain measured 2580 m × 2570 m × 175 m. The size of the domain should be large enough so that the exact positions of the boundaries have no significant effect on conditions in the region of interest. For dispersion applications, such as this one, the requirements for concentration and dose information at significant distances downwind of the source will generally lead to sufficiently large domains as a matter of course.

The mesh comprised 28 × 31 cells in each horizontal layer. The central region was represented by 18 × 21 cells, with lateral cell dimensions varying from about 10 m up to 80 m. In the vertical direction, 15 cells were used, with uniform 5 m thick cells up to 50 m and then expanding upwards for a further 125 m. The various building heights were allocated to lie in 5 m bands. Considering this mesh distribution, there are several unsatisfactory features.

The mesh is very coarse. Large cell sizes will lead to excessive spreading of contaminants and dilution of contaminant concentrations, particularly in the near-source region where the spatial gradients of concentration are large.

The coarse mesh resolution would be inadequate to resolve the velocity and turbulence gradients near the ground and the complex behaviour in the vicinity of the buildings. Recommendations for the minimum numbers of cells needed to ensure 'adequate' modelling of the flows around buildings typically require 10–15 cells over the height of a building, with the smallest cells at the roof edge having a dimension of roughly h/15, where h is the building height (see, for example, Sutton *et al* 1986). It is recognised that this may be difficult to achieve in practice, but it is important to be able to model realistic flow conditions, particularly around the buildings closest to the source.

There are insufficient cells between buildings. The boundary condition at a wall affects the flow conditions in the adjacent cell. There will be a problem if there is only one cell between the walls of two buildings. There should really be some 'free' cells which are not directly modified by the presence of walls.

The expansion of cell sizes is too abrupt. The ratio of adjacent cell dimensions should be kept as close to unity as possible. The upper limit usually recommended for expansion ratios is about 1.2 to 1.3. Large values such as 2 or 3 should be avoided, except in regions where the variables do not change significantly.

The mesh distribution corresponds exactly to the plan dimensions of buildings but not their heights. It would be more consistent to decide on the desired accuracy for representing buildings, eg ±2 m, and then apply this criterion everywhere, but also ensure that the building volume and position are representative.

3.3 Boundary conditions

The atmospheric inflow boundary was located 1000 m upwind of the central region. A uniform velocity profile was specified at the boundary and allowed to develop naturally over the ground. Default 'friction factors' were used for all the solid surfaces, but no details of these friction factors were given and it is assumed that they corresponded to smooth walls. The new boundary layer, driven by the velocity fixed at the top of the domain, would grow upwards from the ground up to the top of the domain, a height of 175 m. In the real atmosphere, a distance of 1000 m would be insufficient for the boundary layer to develop fully over this height. A fetch of over several kilometres would probably be needed to ensure this (Cook 1985). The approach normally taken in wind engineering applications is to specify the boundary layer profiles for the velocities, turbulence properties and temperature at the atmospheric inlet boundaries. An important check which should be carried out is to ensure that the inlet and ground boundary conditions and the atmospheric turbulence model together give equilibrium conditions in the region of interest in the absence of buildings. This is usually done by undertaking a simulation without the buildings present, and checking whether the boundary layer profiles change over the region of interest (see, for example, Hall 1996).

3.4 Sources and concentration calculations

Two types of release were considered. The first of these was a noble gas release and the second was a release of aerosol particles which were assumed to settle at a constant rate. The field concentration units were Bq kg^{-1}. For results purposes, the concentrations were multiplied by the local density to convert them to Bq m^{-3}. In addition, the dose was integrated, with units of Bq s m^{-3}. Evacuee uptake was calculated using the concentrations at the start and end of each time step. A breathing rate of 3.3 10^{-4} m^3 s^{-1} was specified. An evacuation route was specified in terms of specific cells and the times at which the site worker passed through them. Unfortunately, in view of the coarse model resolution, the accuracy of such dose estimates is likely to be poor. Concentrations and doses could be underestimated near the source due to excessive dilution in the large computational cells.

Regarding the release of aerosol particles, the settling of particles was carried out as a post-processing stage at the end of each time step. Cells located immediately above a horizontal surface were assumed to deposit an appropriate proportion of their contents, corresponding to the fixed settling velocity of 10^{-3} m s^{-1} for dry deposition or 10^{-2} m s^{-1} for wet deposition. No account was taken of the local flow behaviour and inertial effects of particles. This approach for wet deposition only removed material at the ground and relied upon diffusion to move material from above into the bottom layer of cells.

3.5 Turbulence modelling

Turbulence was accounted for using a fixed eddy viscosity, $\nu_t = 10^{-3}$ m^2 s^{-1}, compared to a laminar (molecular) viscosity of 10^{-5} m^2 s^{-1}. This differs from the normal approach of calculating the local distribution of eddy viscosity using the k-ϵ turbulence model, say. The assumption of a fixed eddy viscosity is inconsistent with realistic profiles of turbulence and turbulent length scales in the atmosphere and neglects the effects of increased turbulence in the vicinity of buildings, which is particularly important in this type of application. For comparison, in the cube test case described in Section 2.2, the eddy viscosity near the ground and in the vicinity of the building was around 10^{-2} m^2 s^{-1}.

3.6 Other modelling issues

No details are provided in the report regarding the differencing scheme used and it is assumed that it was a first-order accurate upwind differencing scheme. It is important that at least second-order accurate differencing schemes are used to minimise excessive numerical diffusion.

The computational approach for simulating dispersion involved two stages. Firstly, the steady-state wind field was solved for different wind directions, and each of these solutions was stored in a library. In the second stage, the transient dispersion was simulated using the existing steady-state wind field solutions 'linked' together according to a specified time-varying wind pattern. Time steps of 1 hour were used for the concentration calculations. With a wind speed of 5 m s^{-1} and a typical horizontal cell dimension of 50 m, the time step should be around 10 s, which is the time taken for an 'air parcel' to traverse the cell. A time step of 1 hour would give inaccurate results for the development of the concentration field.

3.7 Near-field dispersion examples

The first accident scenario involved a constant release of gas over 50 hours and then a release of aerosol for a period of 3 hours. The specified wind pattern was 5 m s^{-1} from due south veering to 20° west of due south after 2 hours. In the second scenario, the aerosol deposition velocity was increased to 10^{-2} m s^{-1} to represent the effect of rain. The third scenario was the same as the first, but incorporated a southerly 1 m s^{-1} wind throughout the calculation, and a simple representation of an inversion. For the fourth scenario, the wind speed was set at 3 m s^{-1} blowing initially from 10° north of east and veering after 6 hours to blow from 30° south of east. In the last scenario, gas and aerosol releases were specified for a period of 3 hours, with a 5 m s^{-1} wind blowing initially from 30° south of due westerly and shifting to 10° south of due west after 1 hour.

The key feature of these scenarios is the time-varying wind pattern. A time step of 1 hour was used, but as discussed in Section 3.6, time-accurate results would require much shorter time steps. Treatment of a veering wind might then be less satisfactory than as shown in the feasibility study. Either solutions would have to be obtained for more wind directions or large step changes in direction would have to be accepted. The alternative, much costlier approach would be to solve for the time-dependent wind flows as well as for dispersion.

The representation of an inversion by a frictionless ceiling in the third accident scenario is rather crude. Apart from any considerations of the realism of the upwind boundary layer profiles, the ceiling approach would artificially increase the speed of the wind blowing past the site, since no account is taken of the blockage effect of the buildings, which would cause an increase in the height of the flow streamlines over the site.

The results are discussed in some detail in terms of the local flow patterns, concentrations and deposition, although they are obviously dependent on the resolution of the mesh. Whilst a finer mesh would be used in practice, due to the size and complexity of site models and the heavy computational burden of transient calculations, it is extremely likely that the results would still be mesh dependent. Great care would be needed in using and interpreting the results.

Another aspect which needs to be considered is the accuracy of small concentrations and what should be regarded as the working lower threshold. This is a similar problem to instrument accuracy. For example, a gas sensor might have a stated lower limit of resolution of 0.1% for concentrations less than 2%.

3.8 Future modelling improvements

The feasibility study report briefly deals with some aspects which would need to be included in a proper study, namely ground roughness, temperature inversions and radioactive decay. Failure to represent surface roughness adequately will prevent equilibrium boundary layer conditions from being achieved as discussed in Section 3.3. For stable atmospheric conditions, thermal effects would also need to be accounted for, including the heat flux boundary condition at the ground. Both these aspects were considered in some detail in the HSE and EMU projects described in Section 2.

3.9 Validation

Two approaches were suggested for validation. Firstly, it was suggested that if simulations of dispersion around simple geometries were shown to correspond well to field trials, then it may be expected that CFD will also simulate other geometries well. The availability of acceptable datasets is the main problem with this approach. There are several relevant datasets available, but most have some shortcomings for CFD validation purposes. More information is required for validating CFD models than for simple Gaussian plume models, since flow data as well as concentration data are needed. For example, detailed information about the prevailing atmospheric boundary layer conditions is needed, including vertical profiles of velocity and turbulence properties.

The second validation suggestion was that field trials could be undertaken using tracer gases under stable weather conditions. The use of field trials would be preferable to the use of wind tunnel data, due to the potential scaling issues of model-scale experiments, but could prove to be frustrating. The variability of wind conditions and the large instrumentation requirements for a meaningful CFD-orientated data gathering exercise are important issues. If the field trial conditions were not well characterised, then comparison with CFD results would just lead to uncertainty.

3.10 General comments

The feasibility study demonstrated the functional capabilities of CFD for near-field dispersion modelling. The practicability of obtaining realistic and accurate solutions was not proven, since a coarse resolution model was employed.

It could be argued that the accuracy required of CFD simulations for near-field dispersion need only be equivalent to the accuracy currently achieved by analytical flat terrain models. Simpler CFD models might then be justifiable and the computational costs would be much lower. The major problem with such an approach is the current variability of CFD results between different modellers. Significant differences have been shown to occur even for moderately fine meshes. CFD modelling allows far more degrees of freedom than do analytical models and the potential for differences between the results of different modellers, and hence organisations, would be great if coarse resolution models were used.

4 Choice of modelling techniques

This section considers the applications for which analytical models, CFD codes or physical modelling would be most appropriate. For the present purposes, the ADMS model (Carruthers *et al* 1994) has been taken as representative of the capabilities and performance of analytical models. The different methods are discussed below in terms of four key issues: the capability of the method for representing realistic cases, the practicability of carrying out the work, the accuracy which should be achievable, and the uncertainty of the results in practice due to the way in which the technique is used.

4.1 Capability

The capabilities of the three methods are discussed firstly with respect to the modelling of realistic geometries. This is followed by comments on the treatment of non-neutrally buoyant releases, source momentum and time-dependency effects, and, finally, stable atmospheres.

Complex building and structural details are generally easier to reproduce in wind tunnel models than in CFD models, although small-scale and porous features may be difficult to represent accurately in a physical model. The smallest features which can be resolved are about 2–3 mm. If the model scale was 1 : 300, then this would correspond to about 1 m at full-scale. In comparison to the construction of a physical model, the effort involved in setting-up a CFD model is currently quite similar. Generating CFD meshes around arbitrary complex geometries has become much easier with the advent of graphics-orientated mesh generators and further advances in terms of the ease and speed of set-up are possible by using CAD packages. In addition, automatic mesh generation techniques are being implemented in the main commercial CFD codes, which should enable dramatic savings in set-up cost. In the meantime, the mesh generation process can be quite involved when different types of shapes are present in the same model: for example, industrial sites with building blocks, large diameter cylindrical storage tanks, and smaller diameter horizontal axis cylindrical tanks. Difficulties can arise because the CFD mesh has to represent the open space around different objects, rather than the objects themselves. The mesh may be designed easily to fit around one specific building or storage tank, but it also has to interface exactly with other parts of the overall mesh designed to fit other objects. For a large mesh with numerous objects, a considerable amount of cross-referencing is therefore required. Another issue is the ease with which the physical or CFD model can be modified. This may be necessary if there are design changes or if a greater level of geometrical detail is needed. The addition of new features into an existing CFD mesh can sometimes be difficult because of the topology of the mesh: for example, putting in a new building which is orientated at an oblique angle to the existing mesh cells. To avoid this, it helps to know beforehand what features are to be represented.

Complex terrain is represented quite differently in physical and CFD models. Physical models use layers of sheeting cut out according to the contour patterns of terrain height. For applications demanding the greatest accuracy, the step changes in height at the edges of the individual layers can be filled in. The smooth terrain shapes can then be roughened as necessary to give the appropriate ground roughness. This technique is rather time-consuming and is thus not implemented very often. With CFD modelling, the terrain heights are interpolated at the vertices of the computational cells to give a multifaceted representation of the topography. A potential problem may arise when the spot heights at the corners of a cell face result in the vertices being non-coplanar. This 'warpage' introduces errors, rather like truncation errors, due to the cell face having more than one normal vector.

Considering building geometries, analytical models such as ADMS are only capable of representing simple cuboid shapes. The 'building effects module' in ADMS, described by Robins *et al* (1996), defines several regions of flow and dispersion around a cuboid building, eg upwind, recirculating (near-wake) and wake zones. The treatment of realistic, complex buildings requires the user to specify the dominant building within any group, while the code then calculates the effective building. Another point is that the data on which the building effects calculations are based generally correspond to flat terrain, although the software package may combine the building effects calculations with dispersion calculations for complex topography. Regarding calculations of wind flow and dispersion over topography, such as in the FLOWSTAR component of ADMS, the methods are limited to slopes up to about one in three. Sites with large step changes in elevation cannot therefore be represented accurately.

Regarding buoyant or dense gas releases, CFD and analytical models have an advantage over wind tunnel models in being able to use data and models derived from a wide range of experiments including field trials. Full-scale conditions are modelled directly and scaling problems are avoided. Wind tunnel scaling remains an issue for buoyant or dense releases. Complete scaling, involving conservation of the emission density ratio, emission velocity ratio and Froude number, leads to the following requirement for very low wind tunnel operating speeds:

$$U_m/U = (L_m/L)^{1/2} = \epsilon^{1/2}$$

where ϵ is the geometrical scale ratio, U is a characteristic wind speed, and L is a characteristic length scale (suffix m denotes model scale). An alternative approach for scaling is to relax the requirement to conserve the density ratio, instead focusing on conservation of non-dimensional source buoyancy and momentum fluxes. This has been shown by Robins and Obasaju (1996) to be successful for a range of applications, although they stressed the need for the user to demonstrate that the results were independent of the distortions applied to particular parameters.

Where source momentum effects are important, CFD and analytical models have the advantage over wind tunnel modelling. CFD has greater flexibility in terms of release location, orientation and complex interactions, eg with adjacent buildings or other discharges. The shortcomings of physical modelling include practical aspects such as the tubing requirements for achieving a certain discharge flow rate. Time-varying releases are also more complicated to deal with in wind tunnels. Usually, a large number of repetitions have to be undertaken in order to achieve accurate ensemble mean values. For example, around 100–200 releases may be needed in situations where the dispersion is strongly affected by atmospheric turbulence. For certain applications such as dense gas releases, fewer releases may be necessary. The repetition of releases is automated in some wind tunnels and this obviously simplifies the problem.

Stable atmospheres can be dealt with readily by analytical models. Some care is needed with CFD due to the isotropic nature of the k-ϵ model, since the anisotropy of the atmospheric turbulence is accentuated by the stratification. Physical modelling of stable atmospheres requires special wind tunnels, such as that at the EnFlo Research Centre (University of Surrey), capable of supplying the necessary heating and cooling fluxes to achieve the thermal stratification.

Deposition modelling is an area in which little work appears to have been done using CFD models. The feasibility study adopted a constant settling velocity of 10^{-3} m s^{-1} for dry deposition and 10^{-2} m s^{-1} to represent the effects of rain. Local flow effects on deposition were not considered. The approach taken for wet deposition only removed material at the ground and relied upon diffusion to move material from above into the bottom layer of cells. This strategy could be improved upon by introducing a sink term into each cell, irrespective of height above ground, to remove material from the whole plume. Eichorn *et al* (1996) mentioned the implementation of deposition modelling into the MISCAM code, but no details were given. Most (non-CFD) dispersion models use simple, empirical dry deposition velocities. A few models use the more sophisticated resistance analogy approach for dry deposition in which account is taken of the effects of atmospheric turbulence, surface type and particle size. For wet deposition, the standard approach involves empirical scavenging ratios and uniform removal over the whole depth of the plume. Although these deposition modelling techniques could be easily incorporated into CFD codes, they are not really consistent with the approach taken in CFD modelling, because the simple deposition velocities represent global average transport rates, whereas CFD is used to model local conditions. An alternative strategy might be to use a Lagrangian aerosol transport model; most commercial codes have this capability. Provided that the particles have no effect on the flow field, the aerosol transport calculations can be decoupled from the main flow calculations. It may then be feasible to consider calculating the transport of very large numbers of particles. Particle size distributions can be specified and gravitational settling and aerodynamic effects are implicitly taken into account. Blackmore (1996) demonstrated such an approach for modelling precipitation, with calculations of the individual trajectories of up to 900,000 droplets.

4.2 Practicability

The issue of practicability takes account of the staff costs, computing requirements and timescales for modelling an actual site in an appropriately detailed manner. Models such as ADMS have a clear advantage in terms of their low cost and rapid calculations. However, if detailed account needs to be taken of the actual geometries or complex interactions, then the choice of method lies between CFD and wind tunnel modelling. For simpler sites with only a small number of buildings, the cost and timescales of setting up a CFD model will generally be smaller than those for wind tunnel modelling. As the site becomes more complex, the effort required for both approaches may rise towards perhaps 3–4 weeks for a large model.

The number and orientation of the wind directions to be studied is important. With CFD models, most of the mesh generation effort will be focused on the site buildings and immediate surroundings. The outer regions of the mesh, including topographical features only, can be generated and regenerated easily to cater for different wind directions. For example, in the EMU project, some modellers experimented with meshes based on the 'turntable' principle commonplace in wind tunnels. The site was included within a fixed inner turntable mesh and the outer mesh was rotated for different directions. In wind tunnels, when the terrain model extends beyond the edge of the turntable, a physical rearrangement of terrain panels or utilisation of new panels in the wind tunnel will be necessary for large changes in wind direction. Currently, CFD probably has a small advantage in this respect. This advantage will undoubtedly grow as more experience is gained and better mesh generation techniques are developed.

Regarding CFD computing times, an indication of run times was given by Hall (1996) for the test cases described in Section 2.2, using a DEC 3000 AXP Model 600 workstation with 128 Mb of core memory. For smaller, simpler problems such as a cube model with about 32,000 cells and a passive gas release, the run times were less than 1 hour. For the larger, complex models, such as the sample industrial site model, with about 175,000 cells, run times of 34 hours were involved in obtaining the initial steady wind field and a further 10–15 hours of computing per minute of 'real time' during the transient dispersion phase. The steady-state simulations involving a jet release of dense gas required roughly 20 hours of computing.

4.3 Accuracy

Only a relatively small number of near-field dispersion applications have been reported in the open literature and, in view of this, it is not possible to make any comprehensive statements of the accuracy currently attainable with commercial CFD programs. However, the results generally do show good qualitative agreement. The accuracy of CFD modelling, in the absence of numerical errors, is governed largely by the ability of the turbulence model to represent the complex fluid dynamics processes present in the atmosphere. The standard k-ϵ model has been shown many times to have significant shortcomings, such as its overprediction of turbulent kinetic energy, its assumption of isotropic turbulence, and its inability to predict flow separation at the upwind edges of roofs without extremely fine meshes. For dispersion applications, the k-ϵ model seems to overpredict consistently the ground-level centreline concentrations downwind of the source. Extended k-ϵ models and more advanced techniques such as RSM and LES have been shown to perform better in certain cases. Validation for realistic sites (multiple buildings, non-flat terrain, etc), non-neutral stability atmospheres, and downwind distances of up to 1–2 km need to be addressed. Some information will be provided by the EMU project, while the datasets used by Götting *et al* (1995) are also quite relevant, but the general accuracy of CFD codes can only be evaluated properly on the basis of a large number of validation cases. Other good experimental datasets therefore need to be identified, particularly for full-scale trials, and comparisons made between the experimental results and CFD predictions.

Simple models can be used to predict near-field dispersion due to low momentum sources in the vicinity of a single building. For ADMS, comparisons with two experimental studies have shown accuracy to be within a factor of three for such simple cases (Robins *et al* 1996). Further comparisons would be needed to obtain a reliable indication of accuracy, and as the complexity of the scenario increases in terms of site geometry, topography and release conditions, the accuracy would be expected to become increasingly uncertain.

The accuracy of wind tunnel modelling of dispersion depends on a range of factors including model scaling, the nature of the simulated atmospheric boundary layer and the release, and practical issues such as instrumentation. The most straightforward scenarios to model are probably passive releases near buildings in neutral boundary layers, and for those cases, wind tunnel modelling is accurate to better than a factor of two. For stable atmosphere cases, the repeatability of experiments becomes more difficult due to the lower operating speeds, forced by scaling laws, and the problems of maintaining constant cooling water flow rate and temperature. Averaging times increase and compromises inevitably have to be made to ensure practicability.

4.4 Uncertainty due to modellers

Whichever modelling technique is used, the results will probably vary to some extent on the precise way in which the technique is applied. For CFD codes, this issue has been the subject of recent research in the EMU project, which has demonstrated the degree of variability that can occur even within a small group of experienced modellers. The main causes of the variation in results are the different meshes and numerical schemes. Very few, if any, modellers are likely to be able to obtain mesh-independent solutions for near-field dispersion problems in the near future. Faced with constraints on project costs and timescales, a modeller has to make compromises on the sizes of meshes and the distribution of cells. The variability is strongly linked to computing resources, which vary significantly from organisation to organisation.

Regarding wind tunnel modelling, there have been no recent comparison exercises between wind tunnels for dispersion applications, although some efforts are being made to develop standard test cases which each wind tunnel facility could undertake in order to demonstrate capability. There have been comparison exercises focusing on wind loading, ie surface pressures and overall forces, and although these highlighted some significant discrepancies, the results obtained by the different wind tunnel facilities still agreed to within +20% (Cook 1990). For dispersion modelling, the variability may also tend to be lower because of the small number of facilities and the relatively high level of experience of the staff.

For analytical models, such as ADMS, there are far fewer degrees of modelling freedom. The variability between users is therefore likely to be smaller. However, one area in which considerable judgement may be involved is in the choice of the dominant building within a group of buildings.

Another key point with all three modelling techniques is that there is always a danger that the techniques are used beyond their limits of applicability. This probably affects analytical modelling more than the complex methods because the calculations are very quick and easy to undertake, and it may not be possible to use more sophisticated models within the available timescales and costs.

5 Conclusions and recommendations

5.1 Conclusions

(a) The feasibility study provided a useful demonstration of the capabilities of CFD for modelling near-field dispersion and deposition, although it was not really representative of a 'real' application because an extremely coarse resolution model was used and the boundary conditions were inappropriate for simulating realistic atmospheric flows.

(b) Several applications have been reported for large, complex industrial and urban sites, thus confirming the feasibility of using CFD for practical near-field dispersion applications.

(c) The cost and timescales involved in CFD modelling are high compared to those for 'simple' models, but of a similar order to those for wind tunnel testing. CFD probably has a small advantage over wind tunnel testing for larger problems and this advantage will undoubtedly increase as more experience is gained and better mesh generation techniques are developed.

(d) The main benefit provided by CFD, compared to analytical and physical modelling, is the capability to model arbitrary combinations of complex effects, including real site geometries, different stability atmospheres, and buoyant or dense gas releases with significant source momentum effects. There are no scaling problems, and detailed predictions of flows and concentrations are automatically obtained throughout the region of interest, not only at a small number of monitoring locations.

(e) Fine mesh resolution near buildings and sources, accurate numerical schemes and appropriate boundary conditions for CFD modelling of near-field dispersion are important requirements, as demonstrated by recent HSE research and other studies.

(f) Once numerical errors are minimised, the accuracy of CFD predictions becomes limited by the turbulence model. The standard k-ϵ model is used in most cases, but has important shortcomings. There is no consensus regarding modifications to improve the performance of the k-ϵ model. More sophisticated turbulence models, such as RSM, have been shown to provide greater accuracy in certain cases, but have generally not been applied to practical dispersion applications. For the future, LES has great potential but requires substantially greater computing resources.

(g) Validation for realistic sites (multiple buildings, non-flat terrain, etc), non-neutral stability atmospheres, and downwind distances of up to 1–2 km is limited. Some information will be provided by the EMU project, while the datasets used by Götting *et al* (1995) are also quite relevant, but the general accuracy of CFD codes can only be evaluated properly on the basis of a large number of validation cases.

(h) The EC-funded EMU project has demonstrated the variability of CFD results for near-field dispersion due to the way in which a code is used. The main causes of the variation in results are the different meshes and numerical schemes. Available computing power is one of the key issues. This could be a major problem since there is the potential for significant differences between the results of different modellers, and hence different organisations. Some form of sensitivity study should always be undertaken.

(i) The EMU project also highlighted that a 'dispersion expert' still needs to have experience of the specific CFD code being applied to avoid 'mistakes', and that familiarity with a new code takes several months to achieve.

5.2 Recommendations

(a) The variability of results clearly needs to be reduced. One way to address this would be to develop and adopt detailed, strict guidelines (for example, in the form of a voluntary code of practice) covering all aspects of applying CFD to a real site including, in particular, aspects such as mesh design. The adoption of strict criteria might lead to limits on the scale of problem which can be tackled corresponding to the available computing power.

(b) The accuracy of CFD modelling for this application can only be evaluated properly on the basis of a large number of validation cases. Good experimental datasets need to be identified, particularly for full-scale trials, and comparisons made between the CFD and experimental results.

(c) Further work is needed on turbulence modelling for atmospheric dispersion applications. Particular aspects which need to be addressed include anisotropic diffusion, atmospheric stability and building effects. The performance of 'improved' k-ϵ models should be investigated for practical dispersion applications.

(d) Further work is needed to develop and validate appropriate deposition techniques for CFD modelling.

(e) Research and development are continuing in the key areas of automatic mesh generation and numerical error estimators, and will eventually help to eliminate the differences in numerical accuracy. The effectiveness of such measures will need to be evaluated in the context of near-field dispersion.

6 References

Almeida, G P, Durao, B F G, and Heitor, M V (1993). *Experi. Thermal Fluid Sci.*,**7**, 87–101.

Basara, B, and Younis, B A (1992). Progress in the prediction of turbulent wind loading on buildings. *J. Wind Eng. Indust. Aerodynam.*, **41–44**, 2863–74.

Benodekar, R W, Goddard, A J H, and Gosman, A D (1985). Predicting lift-off of major self-heating releases under the influence of a building. Luxembourg, CEC, EUR 9675 EN.

Benodekar, R W, Goddard, A J H, and Gosman, A D (1987). Computer modelling of gaseous effluent dispersal from nuclear reactor installations. CEGB/SERC co-funding project joint report for GR/D/07824 and GR/D/51520.

Blackmore, P A (1996). Modelling precipitation deposition rates using CFD methods. Presented at Wind Engineering Society, 3rd UK Conference on Wind Engineering, Oxford 1996.

Boysan, H F (1993). Renormalization group theory based turbulence models and their applications to industrial problems. Presented at IMechE European Conference on Engineering Applications of CFD, London, September 1993.

Breuer, M, Lakehal, D, and Rodi, W (1995). Flow around a surface mounted cubical obstacle: comparison of LES and RANS results. Presented at IMACS-COST Conference on Computational Fluid Dynamics, 3-D Complex Flows, Lausanne, September 1995.

Carruthers, D J, Holroyd, R J, Hunt, J C R, Weng, W S, Robins, A G, Aplsey, D D, Thompson, D J, and Smith, F B (1994). UK-ADMS: a new approach to modelling dispersion in the earth's atmospheric boundary layer. *J Wind Eng. Ind. Aerodynam.*, **52**, 139–53.

Chen, Y S, and Kim, S W (1987). Computation of turbulent flows using an extended k-ϵ turbulence closure model. NASA CR-179204.

Christolis, M N, Christidou, C A, Boudouvis, A G, and Markatos, N C (1995). Modelling pollutants dispersion around buildings on fire. IN *Air Pollution Theory and Simulation* (Air Pollution III) (H Power, N Moussiopoulos and C A Brebbia, Eds). Southampton, Computer Mechanics.

Cook, N J (1985). *The Designer's Guide to Wind Loading of Building Structures – Part 1: Background, Damage Survey, Wind Data and Structural Classification.* London, Butterworths.

Cook, N J (1990). *The Designer's Guide to Wind Loading of Building Structures – Part 2: Static Structures.* London, Butterworths.

Cowan, I R (1996). A comparison of wind tunnel experiments and computational simulations of dispersion in the environs of buildings. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Cowan, I R, Castro, I P, and Robins, A G (1996). Numerical considerations for simulations of flow and dispersion around buildings. Presented at CWE'96, Fort Collins, Colorado 1996.

Delaunay, D, Flori, J P, and Sacre, C (1996). Numerical modelling of gas dispersion from road tunnels in urban environments: comparison with field experimental data. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Eichorn, J, Beltz, N, and Heil, O (1996). A microscale pollution dispersal model for road traffic related pollution – simulation and measurements. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Freitas, C J (1995). Perspective: selected benchmarks from commercial CFD codes. *Trans. ASME,* **117**, 208–18.

Gilham, S, Deaves, D M, Hall, R C, Lines, I G, Porter, S R, and Carter, D A (1996). Realistic modelling of toxic gas releases for risk assessment. Presented at ESREL'96 – PSAM III, Crete, June 1996.

Goddard, A J H, Gosman, A D, Liu, R, and McGuirk, J J (1992). Numerical modelling of unsteady buoyant gaseous releases from nuclear reactor buildings. Final Report for SERC/ERCOS Grant GR/E 82316.

Götting, J, Winkler, Ch, Rau, M, Moussiopoulos, N, and Ernst, G (1995). Plume dispersion over built-up areas: a comparison of numerical results and wind tunnel studies. IN *Air Pollution Theory and Simulation* (Air Pollution III) (H Power, N Moussiopoulos and C A Brebbia, Eds). Southampton, Computer Mechanics.

Hall, R C (1996). Dispersion of releases of hazardous materials in the vicinity of buildings. Phase II – CFD modelling. London, HSE Research Contract RSU 8000, Report WSA/8013.

Hall, R C, Albergel, A, Bartzis, J G, and Cowan, I R (1996). Uncertainty in modelling of gas releases at industrial sites. Presented at ESREL'96 – PSAM III, Crete, June 1996.

Hatcher, R V, Meroney, R M, Peterka, J A, and Kothari, K (1978). Dispersion in the wake of a model industrial complex. Washington DC, Nuclear Regulatory Commission, Report NUREG-0373.

Jaeschke, W, Trumbach, J V, Ries, R, and Heil, O (1986). Dispersion modelling in the vicinity of tunnel exits – simulation and measurements. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Jicha, M, Katolicky, J, Karki, K C, and Patankar, S V (1996). Modelling of pollutants distribution at city road tunnel. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Kobayashi, M H, Pereira, J C F, and Siqueira, M B B (1994). Numerical study of the turbulent flow over and in a model forest on a 2D hill. *J. Wind Eng. Ind. Aerodynam.*, **53,** 357–74.

Laurence, D, and Mattei, J-D (1993). Current state of computational bluff body aerodynamics. *J. Wind Eng. Ind. Aerodynam.*, **49,** 23–44 (1993).

Leschziner, M A, and Rodi, W (1981). Calculation of annular and twin parallel jets using various discretisation schemes and turbulence model variations. *J. Fluid. Eng.*, **103,** 352–60.

Lines, I G, Hall, R C, Gallagher, P, and Deaves, D M (1994). Dispersion of releases of hazardous materials in the vicinity of buildings. Phase I. London, HSE Research Contract RSU 8000, Report WSA/8006.

McQuaid, J (1976). Some experiments on the structure of stably stratified shear flows. Sheffield, Safety in Mines Research Establishment, Technical Paper P21.

McQuaid, J, and Roebuck, B (1985). Large scale field trials on dense vapour dispersion. Final report to the sponsors on the heavy gas dispersion trials at Thorney Island 1982–84. London, HSE.

Moros, A, Ryall, D, Catherall, M, and Akhurst, M (1992). Flow and dispersion in floating roof oil storage tanks. Presented at Inaugural Conference of Wind Engineering Society, Cambridge, 1992.

Murakami, S (1993). Comparison of various turbulence models applied to a bluff body. *J. Wind Eng. Ind. Aerodynam.*, **46&47**, 21–36.

Murakami, S, Mochida, A, Hayashi, Y, and Sakamoto, S (1992). Numerical study on velocity pressure field and wind forces for bluff bodies by k-ϵ, ASM and LES. *J. Wind Eng. Ind. Aerodynam.*, **41–44,** 2841–52.

Perdikaris, G A, and Mayinger, F (1994). Numerical simulation of heavy gas cloud dispersion within topographically complex terrain. *J. Loss Prev. Process Indust.*, **7**, No. 5.

Richards, P J, and Hoxey, R P (1992). Computational and wind tunnel modelling of mean wind loads on the Silsoe Structures Building. *J. Wind Eng. Ind. Aerodynam.*, **41–44**, 1641–52.

Richards, P J, and Hoxey, R P (1993). Appropriate boundary conditions for computational wind engineering models using the k-ϵ turbulence model. *J. Wind Eng. Ind. Aerodynam.*, **46&47,** 145–53.

Robins, A G, and Castro, I P (1977a). A wind tunnel investigation of plume dispersion in the vicinity of a surface mounted cube – I: the flow field. *Atmos. Environ.*, **11**, 29–71.

Robins, A G, and Castro, I P (1977b). A wind tunnel investigation of plume dispersion in the vicinity of a surface mounted cube – II: the concentration field. *Atmos. Environ.*, **11**, 299–311.

Robins, A G, and Obasaju, E D (1996). Scaling requirements for simulating atmospheric dispersion in a wind tunnel. Presented at WES'96, Wind Engineering Society, Oxford, September 1996.

Robins, A G, Carruthers, D, and McHugh, C (1996). The ADMS building effects module. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Savory, E, Toy, N, and Ahmed, S (1993). Experimental study of a plume in a crossflow. *J. Wind Eng. Ind. Aerodynam.*, **60**, 195–209.

Selvam, R P, and Konduru, P B (1993). Computational and experimental roof corner pressures on the Texas Tech Building. *J. Wind Eng. Ind. Aerodynam.*, **46&47**, 449–54.

Sini, J-F, Anquetin, S, and Mestayer, P G (1996). Pollutant dispersion and thermal effects in urban street canyons. *Atmos. Environ.*, **30**, No.15, 2659–77.

Sutton, S B, Brandt, H, and White, B R (1986). Atmospheric dispersion of heavier-than-air gas near a two-dimensional obstacle. *Boundary Layer Meteorol.*, **35,** 125–53.

Thompson, R S, and Lawson, R E (1990). Terrain ampification factors for sources near complex terrain. Washington DC, Environmental Protection Agency, Data Report, Project ACCD, GEPCPX.

Tinarelli, G, Anfossi, D, Brusasca, G, Ferrero, E, Giostra, U, Morselli, M G, Moussafir, J, Tampieri, F, and Trombetti, F (1994). Lagrangian particle simulation of tracer dispersion in the lee of a schematic two-dimensional hill. *J. Appl. Meteorol.*, **33**, 744–56.

Unnamed Organisation (1995). Near-field dispersion using PHOENICS – a feasibility study.

Zhang, Y Q, Huber, A, Arya, S P S, and Snyder, W H (1993). Simulation to determine the effects of incident wind shear and turbulence level on the flow around a building. *J. Wind Eng. Ind. Aerodynam.*, **46&47,** 129–34.

Zhang, Y Q, Arya, S P, and Snyder, W H (1996). A comparison of numerical and physical modeling of stable atmospheric flow and dispersion around a cubical building. *Atmos. Environ.*, **30**, No. 8, 1327–45.

APPENDIX

Glossary of CFD Terms

This appendix aims to provide some brief definitions for the CFD terms used in the text, since they may not be familiar to everyone working in more broadly based atmospheric dispersion applications.

Algebraic stress modelling ASM is a particular form of approach to turbulent flow calculations which attempts to combine the benefits of two-equation models without the limitations of the eddy viscosity hypothesis, or the complication of solving the full Reynolds stress transport equations.

Boundary conditions The solution of all equations in CFD requires the specification of the behaviour of flow variables at the extremities of the computational domain. These boundary conditions may be implemented in a number of ways depending on the form of boundary condition to be applied, the location of the grid nodes for each variable relative to the boundary and other factors related to the solution scheme or general approach.

Differencing scheme Differencing schemes approximate the derivatives in the conservation equations.

Eddy viscosity (hypothesis) This states that the effect of turbulent eddies on local shear stresses is equivalent to imposing an additional (local) eddy viscosity to the fluid molecular viscosity.

First-order schemes This is the general term given to either finite difference formulae or finite volume approximation functions in which the leading (dominant) error term is of the order of δx, where δx is the local cell dimension. The numerical errors may therefore be significantly larger than in second-order accurate schemes.

Higher-order schemes This is the general term given to either finite difference formulae or finite volume approximation functions aimed at more accurate, stable representations of the convective terms in the Navier-Stokes equations. Such schemes generally require the use of data from more distance nodes in their formulation, and as such require special care in the application of boundary conditions.

Large eddy simulation LES is a simulation technique for the solution of the Navier-Stokes equations in which large-scale turbulent fluctuations are computed directly on the mesh, and small-scale eddies (smaller than the scale of the mesh) are represented with a so-called subgrid scale model.

Renormalisation group theory RNG theory represents a branch of physics and mathematical modelling in which the representation of small-scale, apparently random processes within discrete forms of the conservation equations for a continuum may be formulated. It has been used to reformulate the k-ϵ turbulence model.

Reynolds averaging This is the basis of the formulation of many turbulence models. It is based on the assumption that a turbulent flow can be characterised as having mean components of velocity, pressure, temperature, pollutant concentration, etc, and small, random fluctuations about that mean. Substitution of expressions for the instantaneous velocities into the Navier-Stokes equations gives rise to the so-called RANS (Reynolds Averaged Navier-Stokes) equations, along with further transport equations for the Reynolds stresses.

Reynolds stresses The Reynolds stress tensor is a 3 × 3 symmetrical matrix, each term of which is the mean of the product of two of the fluctuating parts of the velocity components. The gradients of the Reynolds stresses appear in the RANS equations, and represent a loss of fluid momentum due to turbulence.

Second-order schemes This is the general term given to either finite difference formulae or finite volume approximation functions in which the leading (dominant) error term is of the order of $(\delta x)^2$, where δx is the local cell dimension. The numerical errors may therefore be significantly smaller than in first-order accurate schemes.

Structured grids Structured computational grids are characterised by the property that the number of nodes used in each direction on each grid plane is always the same. This structure fits in well with programming data array structures, and is efficient. The main disadvantage is the restriction that computational domains must be (notionally) hexahedral in shape and the need for grid refinement can lead to an excessively high level of mesh resolution in unimportant areas of the flow.

Truncation error The truncation error is (formally) the leading order error term in a finite difference approximation to a partial derivative.

Unstructured grids The term unstructured implies the use of a finite element type mesh, in which the relationship between node and cell numbers is defined explicitly, giving the freedom to fill a computational domain with arbitrarily ordered cells.

ANNEX C

Rise of a Buoyant Plume from a Building Wake

Alan Robins
ENFLO, UNIVERSITY OF SURREY

---

1 Introduction

Many current practical models for building-affected dispersion assume passive plume behaviour. This is often justified on the grounds of being a conservative approach to the calculation of ground-level concentrations, deposition and doses. However, passive dispersion is unlikely to describe actual behaviour at all wind speeds. In the vast majority of situations the emitted buoyancy and momentum flux will become important as the wind speed decreases – in some cases they will be important at all wind speeds. The objective of this report is to review this subject from the point of view of current practical dispersion modelling and likely future developments. The review deals with source heights ranging from ground level to a little above roof level, and both lighter and heavier than air emissions, although excluding the specific dense gas clouds topic of gravity-driven spread and dispersion at the ground.

We begin with some illustrative examples before going on to review current modelling procedures. We then review the physical processes involved and the available experimental and theoretical evidence which might be used to test and develop modelling techniques. Possible approaches are then discussed and promising lines of development identified before we conclude with a number of recommendations to put these into effect.

2 Examples

The assessment of gamma dose due to the argon-41 content of the shield cooling air discharge from Hinkley Point A Nuclear Power Station (Macdonald *et al* 1988) illustrates very well the importance of plume rise from a building wake. The shield cooling air is released from a roof-level stack with significant emission speed and heat content and the plume rise which results plays an important role in the dispersion process. Gas tracer observations revealed zero ground-level concentrations when the wind speed was less than about 5 m s^{-1} and approximately passive dispersion conditions in strong winds. This information was incorporated in a dose calculation, leading to significantly lower dose levels around the site than previously calculated through conventional methods based on passive dispersion.

The rise of plumes from building wakes can also play a very important role in the dispersion and deposition of some accidental emissions, providing much reduced near-site concentrations and doses, particularly in light or moderate wind conditions. Calculations of the consequences of the Chernobyl accident start after the plume rise phase, using observations to provide plume heights (typically in the range from 300 to 1500 m). Heat emission rates were very high and wind speeds moderate (about 5 m s^{-1} at 10 m), so the plume escaped from the building wake and rose freely in the atmosphere. The situation might have been different had there been strong low level winds. Simple criteria for estimating when a plume may lift largely clear of a building wake are given by Fackrell and Robins (1981) and Hall *et al* (1980) and estimates of the Chernobyl heat emission rates by Sorensen (1987). Gale force winds would probably have been required to 'ground' the plume when emissions were at their hottest. Heat release rates for severe nuclear reactor accidents are typically assumed to be in the range from about 10 to 150 MW, generally much smaller than at Chernobyl. These will escape from reactor building wakes in light winds but not in moderate winds, eg for a 100 MW emission the transition will occur at wind speeds between about 6 and 12 m s^{-1}.

Robins (1994) points out that plume rise may be a crucial factor in light wind conditions, even for relatively weak emissions. We can see this, in admittedly a rather naive manner, by considering a standard Gaussian model applied to a ground-level emission (neglecting for the moment the effects of buildings). The NRPB-R91 model (Clarke 1979) predicts the centreline, ground-level concentration to be

$$C = Q/(\pi U \sigma_y \sigma_z)$$

where Q is the source strength, U is a reference wind speed, and σ_y and σ_z are the plume's lateral and vertical dimensions, respectively. There are two components to the lateral spread, one due to turbulence and one due to wind direction unsteadiness, and in light enough winds the latter dominates. We then have

$$\sigma_y \propto (x/U)^{1/2}$$

$$C \propto Q/(U^{1/2}\sigma_z)$$

so concentrations increase as $U^{-1/2}$ for fixed σ_z. However, the atmospheric state is likely to be either unstable or stable in such light winds and σ_z is much less in the latter than the former and we conclude that light wind, stable conditions produce the greatest ground-level concentrations. However, if the plume is elevated then the ground-level concentration distribution becomes:

$$Cz_p^2 \propto 2\sigma_z^2\ Q/(U^{1/2}\sigma_z)\ \exp(-z_p^2/2\sigma_z^2)$$

where z_p is the plume height, and now the unstable case will produce the highest concentrations if

$$(z_p/\sigma_{zstable})^2 > 2\ln(\sigma_{zunstable}/\sigma_{zstable}) \sim 5 - 8$$

where $\sigma_{zstable}$ and $\sigma_{zunstable}$ are vertical spread in stable and unstable conditions, respectively.

We conclude that a modest amount of plume rise, Δz, changes the stability conditions leading to the highest concentrations on the ground as well as greatly reducing the concentrations themselves. Another interesting point to note is that in many light wind cases the plume rise will determine the plume spread (ie self-induced dilution will dominate, see Jones 1983) and since plume rise is inversely proportional to wind speed we find

$$\sigma_z \propto \Delta z \propto 1/U$$

$$C \propto QU^{1/2}$$

All of which goes to show that the highest ground-level concentrations will arise in moderate rather than light winds. Including buildings in the discussion complicates proceedings without affecting the conclusions. Transferring the debate to the evaluation of gamma dose weakens it somewhat, chiefly by moving the threshold for significant reductions in ground-level dose rate to lighter wind speeds and hence greater plume rise.

These examples provide a clear demonstration of the case for providing a practical model for treating building-affected plume rise and dispersion. However, the processes involved are complex, being the interaction between plume dynamics and the disturbed flow around and in the wake of a building, and the solution is unlikely to be simple. There are other, more complicated approaches involving experimentation at model and full scale or computational fluid dynamics (CFD) calculations and these too deserve consideration. The objective then is to evaluate the ability of current models to handle plume rise and dispersion in building wakes and assess the potential for developing or further refining such models. We begin by summarising current methods and their limitations.

3 Current standard approaches – the NRPB-R91 model

Relevant advice for users of the NRPB-R91 set of models is given in Section B3 of Jones (1983). This deals with both source momentum and buoyancy fluxes, discussing their significance in terms of the non-dimensional parameters, F_M and F_B,

$$F_M = \rho_s WQ/\rho_a U_h^2 H^2$$

$$F_B = g'Q/U_h^3 H$$

where

$$Q = AW$$

$$g' = g(\rho_a - \rho_s)/\rho_a$$

and ρ is the density, U_h is the roof-level wind speed, W is the emission speed, A is the source area, H is the building height, g is gravitational acceleration and g' is modified gravity, and the suffices 'a' and 's' denote ambient and source conditions, respectively.

Threshold values are defined for the two parameters such that if neither is exceeded then the emission can be assumed to be passive. The threshold values, determined from wind tunnel experiments, are given as

$$F_M = F_B = 0.05$$

A formula is provided for estimating momentum rise, Δz, from a stack of height h, where $h \geq H$,

$$\begin{aligned} \Delta z &= 3\beta^{1/2}(W/U_h)d \qquad && \text{when } W/U_h > 1 \\ &= 0 && \text{otherwise} \end{aligned}$$

where β is the density ratio, ρ_s/ρ_a, and d is the source diameter, $d = (4A/\pi)^{1/2}$. However, this is a plume rise relative to the mean streamline through the source and hence generally of little value as streamlines are displaced in passing around buildings. Wind tunnel experiments are recommended should it be necessary to evaluate concentrations once either threshold is exceeded. No method of calculation is given for such circumstances.

As will become apparent in Section 5, a simple threshold treatment is a very crude approximation indeed. Taken a step further, the thresholds would become functions of distance downstream of the building, then perhaps building dimensions and orientation, and so on. The thresholds are really screening criteria and as such should perhaps be kept simple. The real problem lies in the absence of a recommended calculation procedure once a threshold is crossed.

We see that for a given emission $F_M \sim 1/U_h^2$ and $F_B \sim 1/U_h^3$, so the threshold criteria will be crossed as the wind speed falls. Put another way, source effects, particularly plume rise, will dominate in light enough winds. Surface concentrations may well then tend towards zero.

4 Physical processes

The motion of a plume departs from that of the mean streamline through the source by virtue of the emission's excess momentum and buoyancy fluxes, both of which create motion relative to the local airflow and in so doing generate mixing between the plume and its environment. This leads to an increase in the mass flux within the plume, dilutes the material originally released, and may also modify the buoyancy force acting on the plume (eg in stratified ambient conditions). Conservation of momentum dictates that the plume's excess speed decreases as a result of the increased mass flux, so that momentum rise is very much a near-source process. The buoyancy force, while it lasts, continuously increases the plume momentum flux. There may be a region of accelerated motion close to the source, although mixing generally dictates that the excess plume speed decreases as the motion proceeds. The motion effectively comes to an end when the coherence of the plume structure is destroyed by ambient turbulence and shear, or an equilibrium level is reached in a stratified ambient condition.

The plume rise theories of Moore and Briggs (see, for example, Jones 1983) model these processes in an analytical fashion and are consequently widely used. However, they are only applicable to standard boundary layer conditions and therefore of little value in treating rise from a building wake. More recent methods are more general in nature (eg the integral model used in ADMS, see Apsley and Robins 1991) and hence offer the prospect of application to the case in hand. These are integral models, simple expressions of the bulk equations of motion and thermodynamics, written in terms of relatively arbitrary external conditions. Beyond this level we have full solutions of the basic equations (with some level of modelling), namely CFD approaches. These are potentially powerful, but extremely time consuming.

We have talked in terms of excess momentum to emphasise the point that we are dealing with motion generated relative to the ambient flow which in the vicinity of a building departs significantly from the simple horizontal flow assumed in the boundary layer upstream. The basic features of flow around buildings are adequately understood (see, for example, Hunt *et al* 1978). Suffice it to say that the flow is displaced around the building, and that there are regions of retarded flow upstream and in the wake, reversed flow in the near-wake and elsewhere, and accelerated flow above and to the sides. Relatively strong streamwise vortices may be created in the flow over the roof which persist downstream and generate important secondary flows. These, together with the mean flow into the decaying wake, continue to affect streamline patterns some distance downstream from the building, the distance depending on the nature of the ambient flow, being greatest in stable conditions and least in unstable. Turbulence levels are also changed and, in particular, the near-wake is a zone where mixing is rapid and the main wake one where levels are higher than ambient, the excess decaying with distance downstream.

So far, we have discussed the emission and the wake in isolation, but we should note that the dynamics of the wake, particularly the recirculating flow region, will be changed once the volume flux from the emission, Q, become significant. The recirculation region can be characterised by its effective volume and residence timescales, which together form a volume flux or mixing rate scale, Q_R. Using expressions for these scales given by Jones (1983) we have, for a cuboid of height H,

$$Q_R/UH^2 \approx 0.2$$

and the condition that the wake behaviour is not changed by the emission is that

$$Q/Q_R \ll 1$$

$$Q/UH^2 \propto 0.2$$

which suggests that wake modification due to emissions is not uncommon.

The point in saying these things is not simply that the flow is very complex but that a flow model must be an essential part of the solution we seek to predict plume rise from building wakes. Empirical models may avoid this but there seems little value in pursuing them as generality will be very elusive indeed. However, they perhaps have some role when used in a very narrow site specific sense, say to translate wind tunnel data into a calculation procedure. Very few practical dispersion models contain building effects flow models but those that do (eg ADMS) are candidates for handling the plume rise problem. At the more complex level, well beyond practical modelling, we have CFD methods and these may deserve serious attention for some applications, with wind tunnel simulations as an alternative.

5 Experimental and theoretical evidence

The critical buoyancy and momentum flux parameters discussed in Section 3 for the onset of significant reductions in ground-level concentration are based on a number of wind tunnel studies. However, the concept is linked to Brigg's (unpublished) lift-off criteria (see Jones 1983) for ground-level buoyant emissions. This is expressed through the parameter B, Brigg's lift-off parameter, defined as

$$B = g'D/u_*^2$$

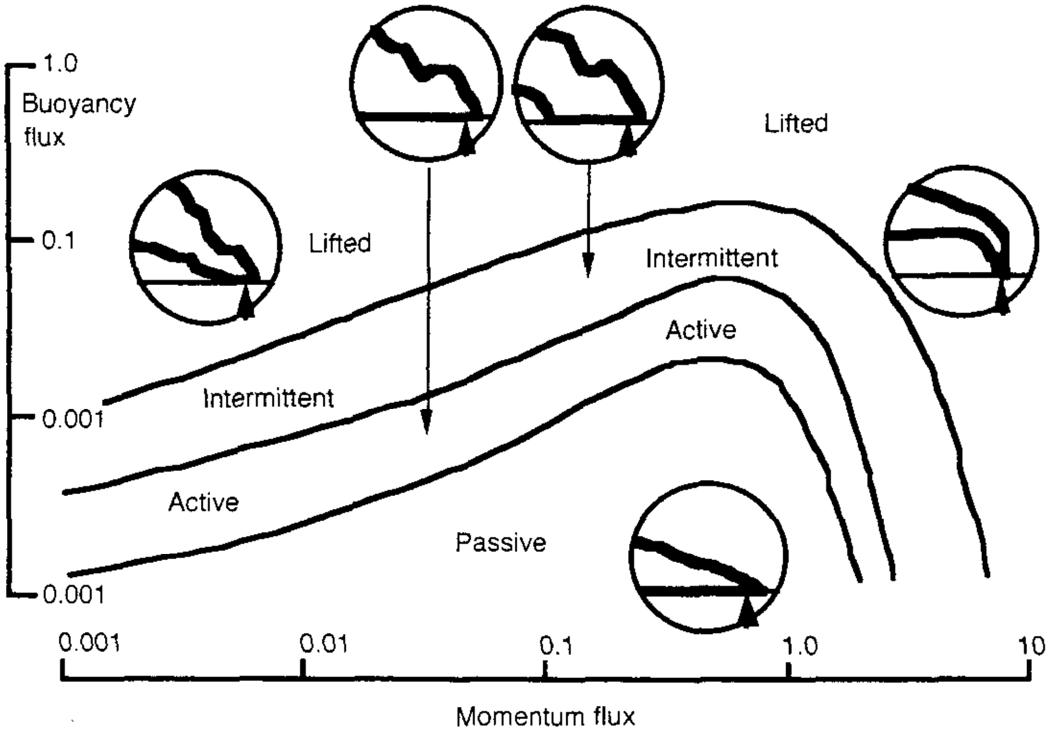
where D is the plume thickness and u_* is the friction velocity. We can see the link with F_B by writing $Q \propto UD^2$ and taking $D = H$, thus giving

$$B \propto g'Q/Uu_*^2H \propto F_B$$

When B exceeds a critical value in the range from 20 to 30 the plume is assumed to lift clear of the ground. However, experiments reveal a rather more complex pattern of events, as illustrated by Figure 1, taken from Robins (1994). Basically, we see intermediate stages of enhanced vertical growth and intermittent lift-off between the passive and fully lifted plume regimes. We can expect a similar picture when a building is present.

There are relatively few experimental studies dealing with the behaviour of non-passive emissions from or near buildings. This contrasts markedly with passive dispersion which has been widely researched and reported; see, for examples, the reviews by Hosker (1984) and Foster and Robins (1985).

Robins and Castro (1977a,b) describe a wind tunnel study of dispersion in the vicinity of a surface-mounted cube. Although primarily concerned with passive or stack emissions, their work contains some information on the effects of source momentum on an emission from the centre of the roof. The effects of emission velocity ratio, W/U_h, in the range from about 0.5 to 6 are discussed, generally with reference to the maximum ground-level concentration. At the 'passive' end of this range the maximum concentration occurs in or near to the recirculation region (say, $x/H < 3$), moving steadily downstream with increasing velocity ratio. The details depend on the orientation of the cube, with the role of emission momentum being most pronounced for the 0° case (ie wind normal to the front face).



The source diameter, d, is used as the characteristic length scale

$F_M = \rho_s W^2/\rho_a U_{ref}^2 \qquad F_B = g'dW/U_{ref}^3$

FIGURE 1 Characteristic behaviour of a ground-level release into a deep turbulent boundary layer as a function of the non-dimensional source momentum and buoyancy fluxes, F_M and F_B

Additional information of this sort can be found in a number of publications, but it is generally of little additional value. This simply reflects the fact that the focus of the research is elsewhere. The work of Koga and Way (1979) deserves mention, even though they were primarily concerned with stack releases. They present a number of simple diagrams showing how plume downwash for roof-level emissions depends on the source location and emission momentum. Ground-level concentration patterns are also plotted.

Fackrell and Robins (1981) undertook a wind tunnel study of the behaviour of a variety of ground- and roof-level emissions from a typical AGR power station. The work addressed such issues as the effects of emission momentum flux, buoyancy flux and duration on mean and fluctuating concentrations in the vicinity of the reactor buildings. Results were analysed to give ground-level, centreline concentration attenuation factors as a function of receptor position and emission characteristics. An attenuation factor, ϵ, is defined as

$$\epsilon(x/H,F_M,F_B,T) = C(x/H,F_M,F_B,T)/C(x/H,0\ 0,\infty)$$

where T is the non-dimensional emission duration, T_sU_H/H, with T_s the emission duration. Each of the source properties was analysed in isolation, so that three separate factors were derived (ϵ_M, ϵ_B and ϵ_T, attenuation factors for emission momentum, buoyancy and duration, respectively); eg for momentum

$$\epsilon_M(x/H,F_M,0,\infty) = C(x/H,F_M,0,\infty)/C(x/H,0,0,\infty)$$

the idea being to relate concentrations in general to the passive emission concentration by writing

$$C(x/H,F_M,F_B,T) = C(x/H,0,0,\infty)\ \epsilon_M(x/H,F_M)\ \epsilon_B(x/H,F_B)\ \epsilon_T(x/H,T)$$

Concentration fluctuation levels in building wakes were included in the study and another factor introduced to estimate maximum short duration concentrations. The mean concentration data were used in setting the threshold buoyancy and momentum flux parameters recommended by Jones (1983).

Fackrell (1983) reports a similar analysis of data obtained during a wind tunnel study of the dispersion of emissions from the then proposed Sizewell B Nuclear Power Station, although in this case only concerned with the effects of emission momentum. Conclusions are rather similar to those reached in the AGR work.

The investigation carried out by Macdonald *et al* (1988) of the dispersion of shield cooling air released from Hinkley Point A Nuclear Power Station has already been mentioned in Section 2. This was a comprehensive piece of research involving gamma-dose monitoring in the vicinity of the site, field studies of the dispersion of chemical and smoke tracers added to the emission, and wind tunnel simulations. Measurements were taken up to 4 km from the point of emission. The work clearly demonstrates the increasing importance of the emission momentum and buoyancy as the wind speed diminishes, with ground-level concentrations eventually falling to zero. An interesting and important point which emerges from the work is that wind tunnel simulations, and for that matter CFD calculations, are likely to indicate a more obviously sharp 'cut-off' than field experiments because of the dependence of lateral spread on wind speed and the likelihood of changes in atmospheric stability in light winds.

In contrast to these CEGB projects dealing with point sources, a number of wind tunnel studies were carried out at the Warren Spring Laboratories concerning the behaviour of area sources of buoyancy and simple block shaped obstacles. Hall *et al* (1980) and Hall and Waters (1986) treat emissions from the walls of buildings, showing how ground-level, centreline concentrations depend on the non-dimensional buoyancy flux, F_B, the source area, A/H^2, and the building geometry and orientation relative to the oncoming wind. Roof, upstream face, downstream face and ground-level sources were investigated. Summary plots are given showing the variation in ground-level concentration at fixed downstream positions with buoyancy flux. The authors emphasise that there is no sudden transition between an entrained and an elevated plume and that ground-level concentrations fall steadily and smoothly with increasing plume buoyancy. These studies were also used in setting the threshold buoyancy and momentum flux parameters recommended by Jones (1983).

More recently, Hall *et al* (1995) report comprehensive wind tunnel simulations of the behaviour of warehouse fires, modelled as releases from area sources in the roof and doorways. Emission buoyancy and momentum were varied in a systematic manner, for a variety of source and building configurations (with $H \sim 10$ m at full scale), and the mean concentration field mapped to downstream distances around 40H–50H. Most attention was focused on ground-level concentrations, with many of the data plotted as non-dimensional concentration $C(x,0,0)U_{10}H^2/Q$ as a function of distance x (m) for ranges of emission and source conditions. Overall behaviour is summarised by sets of figures showing non-dimensional ground-level concentrations at x = 10 and 300 m ($x/H = 1$ and 30) as functions of F_B and F_M, together with plume heights and plume centre concentrations at 300 m, again as functions of F_B and F_M. The report includes numerous flow visualisation pictures. As well as providing an important data source, the work highlights a number of important issues, as follows.

- *Source configuration* – roof emissions from single and multiple vents were studied at fixed overall F_B or F_M. Attenuation of ground-level concentrations is much less for the multiple vent cases than for the single ones.
- *Building shape* – a number of building shapes were studied, differing in plan only and each with a shallow, pitched roof. Streamwise vortices, created by the airflow over the roof, were much less pronounced than commonly found in the wake of sharp edged, block shaped buildings. Consequently, the variation of plume behaviour with wind direction is different.
- *Door emissions* – the behaviour of these emissions was very sensitive to wind direction, a conclusion supported by other investigations of low level emissions (eg during the AGR study reported by Fackrell and Robins 1981).

The subject of the dilution and re-entry of non-buoyant emissions from roof vents has been extensively investigated by Wilson (1985). The work, primarily concerned with the interests of air conditioning and ventilation, deals with surface concentrations on single buildings of various shapes (ie the vents and intakes are on the same building). The analysis is based around wind tunnel studies and the development of simple models for interpolating and extrapolating results.

The downwash of cooling tower plumes has become a topic of considerable interest in recent years with the proliferation of arrays of short cooling tower units at recently constructed gas-fired power stations. Downwash brings the water vapour and droplet laden plumes to ground level, leading to potential fogging and icing problems. One aim of the research has been to determine a critical velocity ratio, W/U_h, for downwash as a function of installation geometry and orientation to the wind. The work has been largely confidential, although the resulting critical wind speeds are applied in some models used to assess these issues for presentation in 'environmental impact' assessments.

CFD simulations of building-affected dispersion initially and quite properly concentrated on passive emissions. Benodekar *et al* (1985) undertook calculations of the Hall *et al* (1980) wind tunnel experiments using a standard turbulence model, but paying particular attention to numerical and grid design questions, although by present-day standards their meshes would be judged to be quite coarse. Agreement with the data was acceptable in the passive emission limit but was at best only modest once source buoyancy effects became significant. Time has moved on and it is now becoming more common for calculations of this sort to be carried out for all types of emissions. Such work is generally associated with site specific safety studies and risk assessments and is not really suited to our purposes, even when it is not confidential. Furthermore, we cannot really claim that grid design and numerical issues have been resolved, or that the optimum level of turbulence modelling has been decided.

Recent research undertaken as part of the European Union Environment Programme (see, for example, Cowan 1996) illustrates the current potential of CFD calculations with the standard k-ϵ turbulence model. A number of test cases have been defined and solved using a commercial CFD system and, in some cases, wind tunnel simulations. Emissions ranged from instantaneous to continuous, and heavier than air to lighter than air, and the surroundings progressed from a single block shaped building to a genuine industrial site. The accuracy of the calculations, where comparisons with data are possible, is best for passive emissions and tends to degrade with increasing plume buoyancy.

6 Modelling approaches

The first level of modelling is to treat all emissions as passive and only pursue something more complex if the overprediction becomes an issue of any kind. This approach can be justified when the passive plume calculation can be relied upon to overestimate actual concentrations. This will generally hold in cases such as

- jet discharges with significant upward momentum,
- buoyant discharges with little or no momentum,

providing the plume motion is not directly affected by impact on to building surfaces, etc. Weak emissions, as defined in Section 3, can also be handled in this way. However, for obvious reasons others cannot be dealt with in this way – the list includes

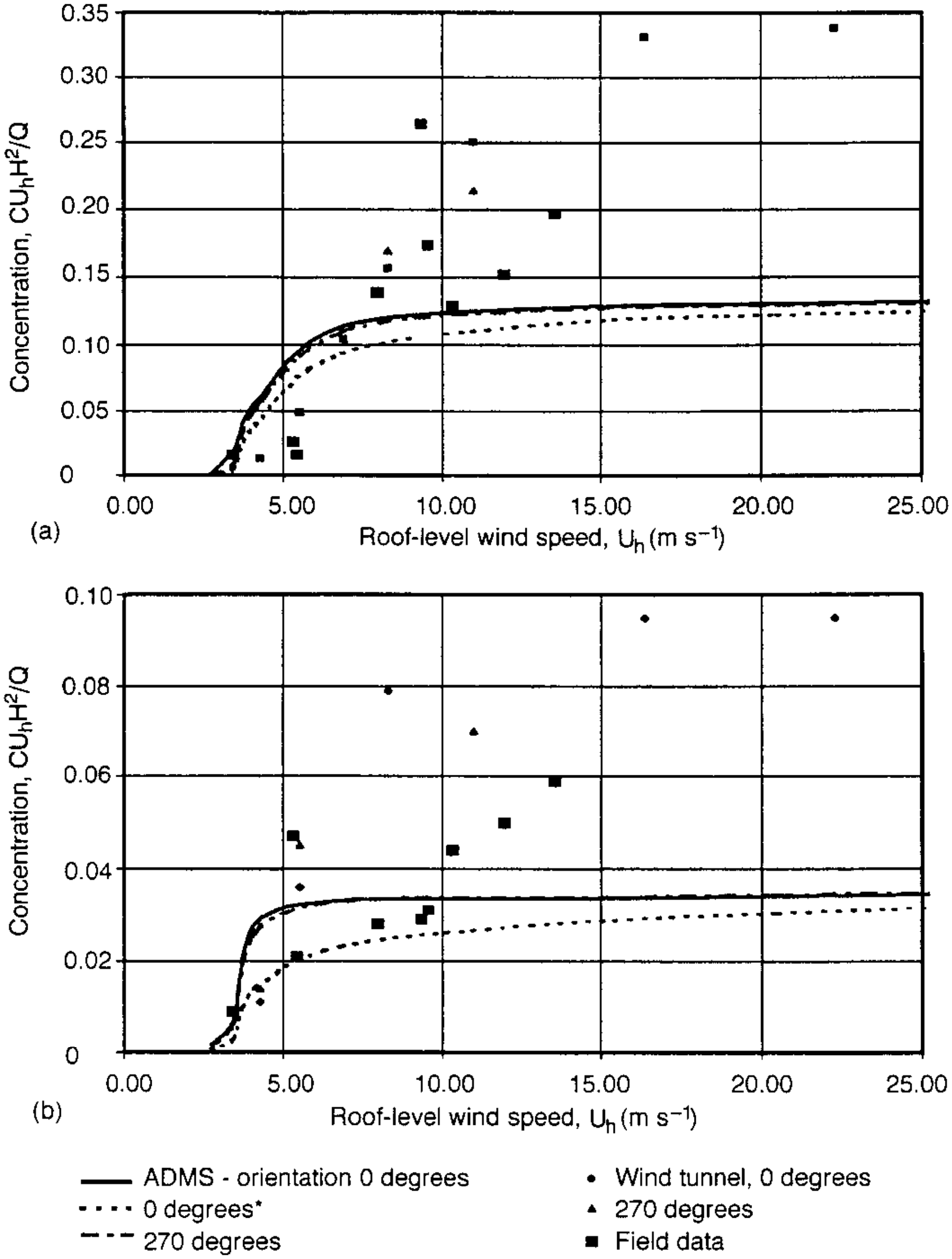
- jet discharges with a significant horizontal momentum component,
- jet discharges with a significant downward momentum component,
- heavier-than-air releases dominated by buoyancy.

The current level of practical modelling has already been discussed. It is generally expressed through critical values of the source parameters, F_M and F_B. At present the recommended calculation procedure ends at this point. Macdonald *et al* (1988) showed that is possible to continue by assuming passive conditions up to a lift-off point and then add plume rise in the method used beyond that point. The lift-off criteria need to be set with the modelling procedure in mind. This is obviously a very crude approximation and cannot be expected to express any detail of the real world – but its virtue is simplicity.

A somewhat more detailed approach is set out by Fackrell and Robins (1981), using attenuation factors which are a function of position. However, this is best viewed as a site specific means of generalising wind tunnel results for inclusion in wider assessments of the consequences of emissions. It has value in this context, but not as a basis for a generalised procedure.

Practical models which contain a reasonably detailed representation of the modifications to flow and dispersion brought about by large buildings, together with an integral model for plume rise, are potentially suitable for the task in hand. One such model is ADMS (see, for example, CERC 1994). Other potential candidates might be further developments of the American EPA model MDNB (Genikhovich and Snyder 1994) or the Russian model OND-86 (Beryland *et al* 1987). To a degree these methods adopt a somewhat similar approach to ADMS, but no detailed evaluation can be made as a full description of MDNB has yet to be published and the OND-86 reference is rather old and in Russian.

ADMS has recently been applied to the Hinkley Point A shield cooling air discharges investigated by Macdonald *et al* (1988). Figure 2 shows predicted centreline, ground-level concentrations as a function of wind speed at 600 m and 2 km from the release point. Concentrations are non-dimensionalised as CUH^2/Q, so that they should become constant in the passive limit or, in other words, at high wind speed. There is a factor of two to three difference between the observed and predicted passive limit concentrations, although that is not the main point to be noted. The important thing is the prediction of a sudden decrease in concentration levels as wind speeds fall from about 6 and 4 m s^{-1}, and zero concentrations in lighter winds. This agrees very well with the data (both field and wind tunnel), although there are differences of detail. No increase in wind speed or other wind speed change over or around the building is modelled and this will certainly be responsible for some of the differences.



*Includes wind direction unsteadiness in calculation

FIGURE 2 ADMS v1.5 predictions and measured non-dimensional ground-level concentrations of shield cooling air discharges at (a) 600 m and (b) 2 km downstream from Hinkley Point A Power Station as a function of wind speed

We noted in Section 4 that the wake dynamics may be changed by the emission once the inequality

$$Q/UH^2 \ll 0.2$$

is not satisfied. We can write this in terms of F_B and F_M as

$$Q/UH^2 = (F_M/\beta)(U_h/W) \ll 0.2$$

$$Q/UH^2 = F_B(U_h{}^2/g'H) \ll 0.2$$

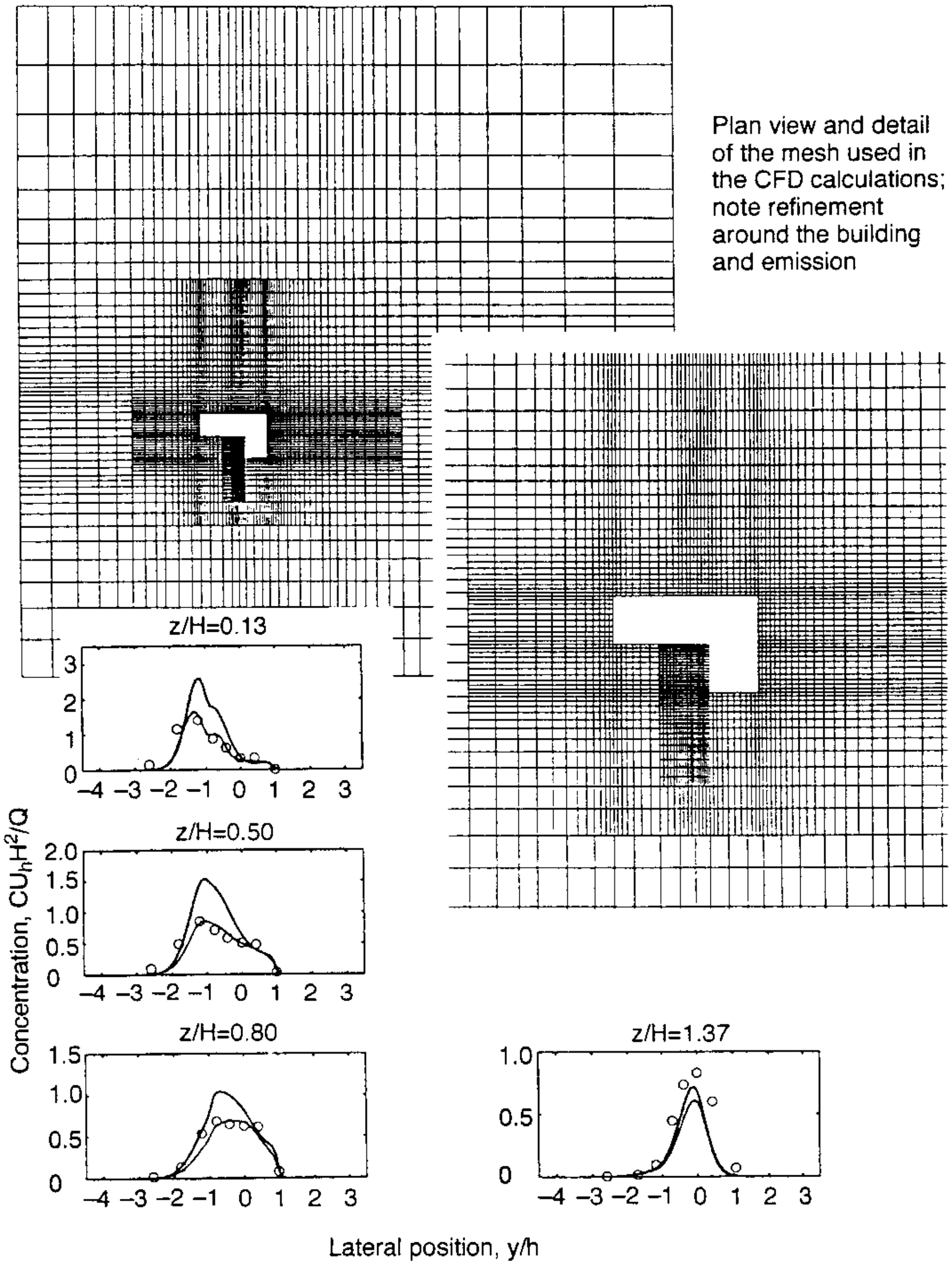
where β is the density ratio, ρ_s/ρ_a. The clear implication is that in many cases the emission will alter the wake flow. There is no way of treating this within models such as ADMS, so we must expect some imprecision to arise as a consequence. This may or may not be a significant issue – only model evaluation will decide.

ADMS modelling is likely to be most successful for emissions at or near roof level. These are much less susceptible to details of building geometry and orientation than, say, ground-level discharges. The latter are sometimes upwind of the building, sometimes to the side and sometimes downwind and behave quite differently as a result. Small-scale building or site details near to the source may deflect the plume or the local mean flow, introducing important dispersion features at a scale well below that which the model can be expected to handle. Clearly, less can be expected of the model in such circumstances.

The ADMS plume rise model assumes that the plume boundaries are clear of the ground and, at the moment, that the emission is directed vertically upwards. The treatment of gradual lift-off from a wake, increased vertical spread due to emission buoyancy, or sideways-directed discharges implies something more sophisticated. Progress is possible in these areas. There are some potential difficulties, chiefly relating to cases where plumes may be blown back on themselves and the plume trajectory model (ie the developed plume rise model) becomes invalid. However, these could be quite easily trapped and treated in some other way, so there is no fundamental reason why ADMS cannot be developed for general application to emissions from buildings. How well it might perform is another question.

We have already commented that CFD models are increasingly used by the safety assessment community to calculate the dispersion of hazardous materials near buildings. Figure 3 (from Cowan 1996) compares wind tunnel measurements and CFD calculations of concentrations downwind of a buoyant jet emission from the wall of an L-shaped building. The agreement between the two is certainly adequate but we should note that the solution is not a final one. This means that the solution is not yet independent of the grid on which it is calculated, or for that matter the turbulence model employed in its calculation. There is no guarantee that the 'true' solution will be 'better' than the one shown. With that said, we should note that the calculations used a standard two-equation turbulence model, a second-order spatial differencing scheme, and a carefully designed, locally refined mesh with approximately 120,000 cells. The calculation took about a day to run on a mid-range Silicon Graphics workstation. A more complex simulation, involving a heavier-than-air jet from the L-shaped building impinging on a tall downwind building and using 150,000 cells, was less successful.

There are a few more comments to be made about CFD solutions. What might be termed 'practical CFD modelling' is almost invariably based on two-equation turbulence models (usually k-ϵ,



Plan view and detail of the mesh used in the CFD calculations; note refinement around the building and emission

$h = 10$ m, $d = 1$ m, $Q = 19.6\ m^3\ s^{-1}$, $\rho_s/\rho_a = 0.55$,

Neutral atmosphere, $U_{10} = 5\ m\ s^{-1}$

FIGURE 3 CFD calculations of the dispersion of a buoyant, horizontal emission from the wall of an L-shaped building compared with wind tunnel data at 0.5h downstream of the building (the solid lines refer to calculations with different grids; symbols are data)

or something even simpler). Most researchers would recommend more complex models, capable of predicting the individual turbulence components, although these are considerably more demanding of computer resources. Some pertinent deficiencies in k-ϵ modelling are

- the turbulent viscosity and diffusivity are scalar, so the vertical and lateral diffusivities must be set equal to or, at best, proportional to each other,
- excess turbulence energy is created in some regions of the flow, notably the upstream faces of buildings,
- the response to buoyancy forces is not well simulated,
- the wall boundary conditions are not treated in a very sophisticated manner.

Nevertheless, the k-ϵ model, sometimes modified to reduce its deficiencies to some degree, remains the practical standard.

That leaves something to be said briefly about wind tunnel modelling, or more broadly physical modelling. This is actually beyond of the scope of the present review and all we need to note is that there are three uses to be considered:

- fundamental studies of specific physical processes,
- generic studies of issues affecting a range of practical problems,
- site specific or otherwise problem specific simulations.

In the first two cases the aim is chiefly to improve the basis, range and validity of practical models. The third case is an alternative to practical modelling where the issues become too complicated and too significant to handle in other ways. Even so, the results will normally be used in conjunction with such a model to complete an assessment procedure. We should concentrate on the first two when considering the development of practical models. The experimental studies summarised in this review provide data at the second level. These are probably not sufficient, chiefly because the most extensive (Hall *et al* 1995) deals with inappropriate building geometry and they all are effectively restricted to neutral atmospheric conditions. We probably also need more at the first level (eg exploring wake structure and development) to support underlying theoretical developments.

7 Further development

We have discussed a number of levels of modelling and illustrated where development might proceed with reasonable expectation of success. These are summarised below.

7.1 Extension to current methods

The methods used by Macdonald *et al* (1988) deserve some further consideration. Their virtue is their simplicity, although they might well have limited applicability as a result. Definitions for the lift-off parameters have to be sought which are optimum with respect to whole calculation procedures. This suggests that they will not be universal, so we need to establish if their definition can be expressed in a relatively straightforward way, or if it will it remain a matter for 'expert' judgement.

7.2 Development of practical models such as ADMS

There are good grounds for expecting this to be fruitful. We need first to evaluate the strengths and weaknesses of the current ADMS model by applying it to the data summarised in Section 4. A development programme can then be drawn up and put in place, followed by a re-evaluation of predictive ability. We can expect this to include the extension of the plume rise model to cover arbitrarily directed discharges and ground-based plumes, modifications to treat low level emissions according to wind direction and source location, inclusion of a simple treatment of the wind field above and to the side of the building and some guidance on the possibility of small-scale site features significantly affecting dispersion behaviour. As noted in Section 6, roof-level releases are likely to be handled more realistically than ground-level ones because they are generally much less sensitive to details of building geometry and orientation.

7.3 CFD procedures

These will not provide a 'practical' model but they might well be used to address very specific questions as an alternative to wind tunnel testing, although at the moment this cannot be done reliably. At this stage, we would probably consider only the k-ϵ model, although almost certainly with modifications to overcome some of the deficiencies noted above. Progress with more advanced models needs also to be monitored. The performance of CFD procedures in the current area of interest has not been determined and an important exercise would be to resolve this. The other essential is a clear and concise statement of what constitutes good practice in CFD work as it is all too easy to obtain impressive looking solutions which are for one reason or another of little value. Here, guidance should cover, amongst other things, turbulence models, grid generation and resolution, domain size, differencing schemes, and boundary conditions.

7.4 Data availability

We have a reasonable body of data, although it is by no means complete. The chief weaknesses are likely to concern simulations of cases of current interest, although we first need to be clear on what these are, before deciding what to do. Nearly all existing studies deal only with the mean concentration field. Model development is critically linked to adequate prediction of the flow field and we clearly need studies where this is provided along with the concentration field. Concentration fluctuations may be important for some applications and data are sparse here as well. Existing information is almost exclusively confined to the neutral atmosphere so there is good cause for extending this into stable and unstable conditions. As far as field data are concerned, we have the Hinkley experiment which might be judged insufficient for validation purposes.

CFD calculations are sometimes put forward as a relatively cheap means of obtaining data. We should be very careful not to accept this without considerable qualification. We know that current methods fail to predict certain common flows adequately. This is usually countered by claiming that CFD can nonetheless be used to illustrate, indeed quantify, the sensitivity of a given phenomenon to changes in initial and boundary conditions, etc. This is sometimes true and equally sometimes not true. The simple message is that we need to tread very carefully and judge each case on its merits.

8 Conclusions and recommendations

(a) The NRPB-R91 model does not specify a calculation procedure for plumes whose dispersion is affected by their initial momentum and buoyancy fluxes – it simply defines when this is likely to occur. Simple extensions based on a sudden switch from passive to lifted behaviour are possible, although they may well be somewhat site specific or emission specific in nature. This approach should be quite quick to develop and implement, but its limitations may well prove to be too great.

(b) The ADMS model already contains some of the capabilities sought and offers a good platform for further development. We first need to discover how well the existing model performs and decide which are the most important areas to improve. An extensive development programme will almost certainly follow, but the prospect of successful completion is high. Even so, we should expect better performance with discharges at or near roof level rather than ground level. Development of the American EPA MDNB model and the Russian OND-86 model should be monitored. Closer collaboration between model developers should perhaps be encouraged.

(c) The capabilities of CFD modelling with the standard k-ε model (and its derivatives) should be assessed and the requirements of good practice made clear. Progress with more advanced models should be monitored. CFD methods do not currently provide practical models – the computational effort is too great – but they will be increasingly used. The implications of their use should be considered in detail.

(d) Existing data would benefit from detailed evaluation and consolidation into a single electronic database. This will make access easier and reliability more certain.

(e) A reasonably comprehensive set of test cases should be defined for which adequate data are available. Model evaluation and development rests on velocity field data as well as dispersion data. Gaps in the available sets of data should be filled if it is judged cost efficient so to do. This process can begin once the key deficiencies are made clear and a programme specifying the cost and timescales for their elimination put forward.

9 References

Apsley, D D, and Robins, A G (1991). Plume rise model specification. Cambridge Environmental Research Consultants, UK-ADMS Paper P11/02C/91, ADMS Technical Specification, June 1991.

Apsley, D D, and Robins, A G (1994). Modelling of building effect in UK-ADMS. Cambridge Environmental Research Consultants, UK-ADMS Paper P16/01K/94, ADMS Technical Specification, June 1994.

Benodekar, R W, Goddard, A J H, and Gosman, A D (1985). Predicting lift-off of major self-heating releases under the influence of a building. Luxembourg, CEC, EUR 9675 EN.

Beryland, M E, Gasilina, N K, Genikhovich, E L, Onikul, R I, and Glukharev, R I (1987). Method of calculation of concentrations of air pollutants the industrial emission contains. Gidrometeoizdat (Hydrometeorological Publishing House), USSR Normative Document OND-86 (in Russian).

CERC (1994). Cambridge Environmental Research Consultants, ADMS1.06 Technical Specification, June 1994.

Clarke, R H (1979). The first report of a Working Group on Atmospheric Dispersion: a model for short and medium range dispersion of radionuclides released to the atmosphere. Harwell, NRPB-R91 (London, SO).

Cowan, I R (1996). A comparison of wind tunnel experiments and computational simulations of dispersion in the vicinity of buildings. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Fackrell, J E (1983). Wind tunnel tests of dispersion from the proposed PWR. Ratcliffe on Soar, PowerGen, CEGB Report TPRD/M/1289/R82.

Fackrell, J E, and Robins, A G (1981). Dispersion of emissions at an AGR site. Ratcliffe on Soar, PowerGen, CEGB Report TPRD/M/1172/R81.

Foster, P M, and Robins, A G (1985). The effects of buildings on low level atmospheric discharges. Luxembourg, CEC, EUR 9980 EN.

Genikhovich, E L, and Snyder, W H (1994). A new mathematical model of pollutant dispersion near a building. Presented at 8th AMS Conference on Applied Air Pollution Modelling Meteorology, Nashville, January 1994.

Hall, D J, and Waters, R A (1986). Further experiments on a buoyant emission from a building. Warren Spring Laboratory, Report LR567(AP).

Hall, D J, Barrett, C F, and Simmonds, A C (1980). Wind tunnel model experiments on a buoyant emission from a building. Warren Spring Laboratory, Report LR355(AP).

Hall, D J, Kukadia, V, Walker, S, and Marsland, G W (1995). Plume dispersion from chemical warehouse fires. Garston, Building Research Establishment, BRE Client Report CR 56/95.

Hosker, R P (1984). Flow and diffusion near obstacles. IN Atmospheric Science and Power Production (D Randerson, Ed). Oak Ridge TN, Technical Information Center, US Department of Energy, DOE/TIC 27601.

Hunt, J C R, Abel, C J, Peterka, J A, and Woo, H (1978). Kinematic studies of the flows around free or surface mounted obstacles; applying topology to flow visualisation. *J. Fluid. Mech.*, **86**, No. 1, 179–200.

Jones, J A (1983). The fifth report of a Working Group on Atmospheric Dispersion: models to allow for the effects of coastal sites, plume rise and buildings on dispersion of radionuclides and guidance on the value of deposition velocity and washout coefficients. Chilton, NRPB-R157 (London, SO).

Koga, D J, and Way, J L (1979). Effects of stack height and position on the dispersion of pollutants in building wakes. Illinois, Institute of Technology, Fluids and Heat Transfer Report R79-2.

Macdonald, H F, Foster, P M, Robins, A G, and Thompson, I M G (1988). Improved estimates of external gamma dose rates in the environs of Hinkley Point Power Station. Berkeley, Nuclear Electric, CEGB Report RD/B/6027/R88.

Robins, A G (1994). Flow and dispersion around buildings in light wind conditions. IN *Stably Stratified Flows: Flow and Dispersion over Topography* (I P Castro and N J Rockliff, Eds). Oxford, Clarendon Press, pp 325–58.

Robins, A G and Castro, I P (1977a). A wind tunnel investigation of plume dispersion in the vicinity of a surface mounted cube – I: the flow field. *Atmos. Environ.*, **11**, 291–7.

Robins, A G, and Castro, I P (1977b). A wind tunnel investigation of plume dispersion in the vicinity of a surface mounted cube – II: the concentration field. *Atmos. Environ.*, **11**, 299–311.

Robins, A G, Carruthers, D J, and McHugh, C (1996). The ADMS building effects module. Presented at 4th Workshop on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Ostende, May 1996.

Sorensen, B (1987). Chernobyl accident: assessing the data. *Nucl. Saf.*, **28**, No. 4, 443–7.

Wilson, D J (1985). Influence of exhaust velocity and wind incidence angle on dilution from roof vents. *ASHRAE Trans.*, **91**, No. 2, Paper HI-85-34 2.

APPENDIX

Nomenclature

A	source area
B	Brigg's lift-off parameter, $g'D/u_*^2$
C	mean concentration
D	plume thickness
d	source diameter, $(4A/\pi)^{1/2}$
F_M	momentum flux, $\rho_s WQ/\rho_a U_h^2 H^2$
F_B	buoyancy flux, $g'Q/U_h^3 H$
g	gravitational acceleration
g'	modified gravity, $g(\rho_a - \rho_s)/\rho_a$
H	building height
h	stack height
Q	source strength, volume release rate
Q_R	recirculation region flux (mixing rate) scale
T	non-dimensional emission duration, $T_s U_h/H$
T_s	emission duration
U	mean wind speed
U_{10}	ten metre wind speed
U_h	roof-level wind speed
U_{ref}	reference speed at edge of boundary layer
u*	friction velocity
W	emission speed
x, y, z	standard coordinate system
z_p	plume height
β	density ratio, ρ_s/ρ_a
Δz	plume rise
ϵ	attenuation factor
ϵ_M	attenuation factor for emission momentum
ϵ_B	attenuation factor for emission buoyancy
ϵ_T	attenuation factor for emission duration
σ_y	lateral plume dimension
σ_z	vertical plume dimension
$\sigma_{zunstable}$	vertical spread in unstable conditions
$\sigma_{zstable}$	vertical spread in stable conditions
ρ_a	ambient density
ρ_s	plume density

On sale through the Stationery Office
£10.00

Stationery Office Publications Centre
PO Box 276, London SW8 5DT

Stationery Office Bookshops
49 High Holborn, London WC1V 6HB
71 Lothian Road, Edinburgh EH3 9AZ
16 Arthur Street, Belfast BT1 4GD
The Stationery Office Oriel Bookshop,
The Friary, Cardiff CF1 4AA
9–21 Princess Street, Manchester M60 8AS
68–69 Bull Street, Birmingham B4 6AD
33 Wine Street, Bristol BS1 2BQ

Also available through booksellers

ISBN 0 85951 404 8