**6.5**

# BUILDING AFFECTED DISPERSION: DEVELOPMENT AND INITIAL PERFORMANCE OF A NEW LAGRANGIAN PARTICLE MODEL

Matthew Hort*, Benjamin Devenish and David Thomson
Met Office, Bracknell, Berkshire U.K.

## 1. INTRODUCTION

The dispersion of pollution released from near building sources is of considerable interest and importance with regard to health and safety and air quality concerns. It is also a very complicated problem being dependent not only upon the meteorological and source conditions but also on the building shape, source to building separation and relative position, and the angle of incidence between the mean wind and building.

Significant experimental research exists dealing with flow and dispersion around and downwind of bluff obstacles submerged in a deep turbulent boundary layer. The large majority of this is concerned with cuboids of varying aspect ratio e.g., Robins and Castro (1977a, 1977b), Li and Meroney (1983a, 1983b) and Snyder (1994). This has resulted in a considerable body of knowledge regarding the flow around cubic buildings and the associated dispersive characteristics of their wakes for many different release conditions. However, there are still significant areas where our knowledge is not detailed enough to fully parametrize the complicated flow characteristics even for a simple cuboid building. Unfortunately this level of complication also means that fully resolved computational 'solutions' are simply not practical for emergency response, environmental impact and/or risk assessment studies. These facts have lead by necessity to the development of empirically based and idealized models that, whilst not representing explicitly all of the physics, do capture certain features that enable 'adequate' prediction of the resulting dispersion. This less than 'perfect' approach has the benefit of being computationally cheap and relatively easy to specify in comparison to the full numerical solutions.

The atmospheric dispersion group of the Met Office (UK) has undertaken to develop the capability of simulating the flow field in the vicinity of an isolated cubical building and subsequently the resulting dispersion. This is based on a Lagrangian stochastic modelling approach and will be fully integrated with the Met Office's dispersion model NAME.

## 2. MODEL FORMULATION

The model is of Lagrangian stochastic type and consists of three parts. The first is the formulation of the mean flow field, the second is the calculation of turbulent velocity statistics and the third is the use of a Langevin equation to simulate the particle trajectory.

### 2.1 Boundary Layer

The model simulates a neutrally stratified boundary layer. The mean velocity is given by $U = (u_*/k)\log((z+z_0)/z_0)$ while the turbulent velocity variances are

$$\sigma_u^2 = 6.25u_*^2\left(1-\frac{4}{5}\frac{z}{\delta}\right) \qquad (1)$$

$$\sigma_v^2 = 4.00u_*^2\left(1-\frac{4}{5}\frac{z}{\delta}\right) \qquad (2)$$

$$\sigma_w^2 = 1.69u_*^2\left(1-\frac{4}{5}\frac{z}{\delta}\right) \qquad (3)$$

where $\delta$ is the boundary layer depth, $u_*$ the friction velocity, $z_0$ the roughness length and $k$ the von-Karman constant set equal to 0.4. We assume that the correlation time scale $\tau$ is the same for the three non-zero correlations $(uu)$, $(vv)$ and $(ww)$ and is expressed as

$$\tau = \frac{kz}{1.3u_*}. \qquad (4)$$

### 2.2 Building Effects

The model treats many of the features of dispersion near buildings. These are: upwind and downwind recirculating regions; streamline deflection and modified mean velocities around the building and within its wake; and modified lateral and vertical mean velocities due the vortex formation for non-normally aligned buildings.

The mean flow field is treated in two parts by distinguishing between the bluff body mean flow and the induced vortex flow, the latter being present for non-normal building to mean wind orientations.

The bluff body mean flow is calculated in two stages; initially the streamlines are defined, as discussed latter, and then incompressibility is satisfied by adjusting the flow speed along the streamlines. The flow field is required to return to its undisturbed upstream state at the end of the building wake region. To describe the parametrization of the flow we will consider just the central vertical plane i.e., a 2-D slice as illustrated in Figure 1. The flow field is defined by one streamline. This streamline, highlighted in Figure 1, divides the flow into an inner region i.e., that part next to the building, and an outer region i.e., that outside of this defining streamline. This streamline is defined explicitly with the other streamlines represented by a transition between the respective boundaries and the defining streamline. Here the 'inner' boundary is the ground, building and recirculation regions and the 'outer' boundary is the limit of

---

*Corresponding author address: Matthew C. Hort, Met Office, Bracknell, Berkshire, RG12 2SZ, U.K.; email: matthew.hort@metoffice.com

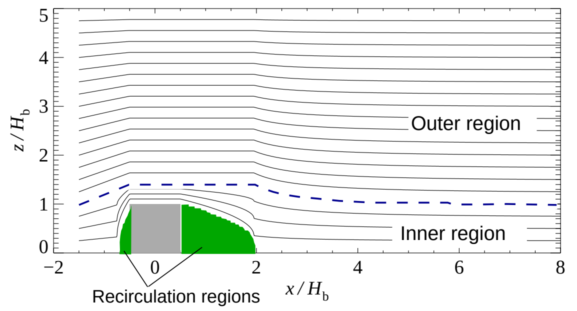

Figure 1: Bluff body mean flow. streamlines shown in $x - z$ plane at $y = 0$.

the computational domain. This process occurs in three dimensions within the model.

The upstream mean flow is defined rather simplistically. The defining streamline has a height of $H_b$ which is increased to $\sqrt{2}H_b$ by the leading edge of the building. The transition being linear from the upwind edge of the computational domain. For positions downstream of the building recirculation region the mean flow is defined using the 3D small-deficit wake model of Robins and Apsley (2000) which is based on the 2D wake model of Counihan, Hunt and Jackson (1972). This directly provides the solution for $U$, while the expressions for $V$ and $W$ are derived from this so as to satisfy continuity. This formulation provides a velocity deficit that decays with increasing downstream distance. We adjust the dividing streamline height so that a measure of the bulk velocity deficit across the inner region matches that predicted by the Robins and Apsley (2000) formulation. The exact formulation has been modified here so that the deficit decays completely by the end of the building wake region rather than as $x \to \infty$.

The so called vortex wake applies to the vortices present in the wake structure downwind of buildings when the incident mean wind is non normal to the building. These are represented using a Combined Rankine Vortex (CRV) formulation. Within this model we consider a vortex solely consisting of a tangential velocity component. The modeled flow in the vortices is characterized by two flow regimes. The first is the vortex core, of radius $r_c$, where the tangential velocity increases linearly with radius $r$, from zero at the centre of the vortex, to its maximum value $V_t$ at $r_c$. In the second regime $r > r_c$ the tangential velocity decreases inversely with $r$. Physically this describes a core in solid body rotation that drives the surrounding fluid at constant angular momentum. Within the model the total vorticity is conserved throughout the vortices downwind evolution. The vortex system is modelled as a set of 4 vorticies, consisting of 2 'real' and 2 image vortices. The latter pair are the mirror image of the 'real' vorticies reflected about the ground. This ensures that fluid is not advected into the ground.

Currently there is no published work in which the characteristics, such as velocity, spatial and temporal variation, of building roof top vortices are dealt with in detail. The simple model developed here has therefore been parametrized through evaluation of the resulting concentration fields for the cubic building case at 45° only.

The building recirculation regions are defined as volumes within which the mean flow is set to zero. The turbulence levels are related to building aspect ratio to incorporate the effect of the mean transport that we do not explicitly represent. The regions have an elliptic boundary. The stream wise extent of the upwind region is currently arbitrarily defined as $H_b/4$. However the extent of the downwind region and the occurrence of leading or trailing edge separation follow from Fackrell (1984).

## 3. NUMERICAL/EXPERIMENTAL SET-UP

Comparison with experimental laboratory data of Hort and Robins (2001) and Robins (2001, private communication) has been conducted as part of the model development and evaluation. All the experimental work was conducted in the environmental wind tunnel at EnFlo, University of Surrey, U.K. The wind tunnel has a working section of 20 × 3.5 × 1.5 m (length × width × height) and can be used to simulate neutral, moderately unstable and moderately stable atmospheric boundary layers.

The numerically simulated neutral boundary layer was defined, based on a 1:400 scaling of the experimental boundary layer reported by Hort and Robins (2001), with a depth $\delta = 400$ m, a roughness length $z_0 = 0.4$ m and a friction velocity $u_*/U_{ref} = 0.058$ where $U_{ref}$ is the velocity at the top of the boundary layer.

Table 1 lists the building and source position configurations considered here. The buildings have height $H_b$, width $W_b$ and length $L_b$. Source height is give as multiples of $H_b$.

| Building Geometry | Orientation | Source Height ($H_b$) |
|---|---|---|
| $H_b = W_b = L_b$ | 0 & 45° | 1, 1.12, 1.25, 1.37, 1.5 |
| $H_b = W_b/2 = L_b/2$ | 0 & 45° | 1, 1.25, 1.5 |
| $H_b = 3W_b = 3L_b$ | 0 & 45° | 1 |

Table 1: Building and release configurations.

In all instances, both numerically and experimentally, the release, positioned over the building $x - y$ centre, was neutrally buoyant with zero vertical momentum. Within this work both numerical and experimental measured concentrations $C_m$ have been converted to normalized concentrations $C$ using

$$C = \frac{C_m U H_b^2}{Q C_s} \tag{5}$$

where $H_b$ (m) is the reference length scale taken to be the height of the building; $U$ (ms$^{-1}$) is the reference velocity taken to be the velocity at height $H_b$; $Q$ (m$^3$s$^{-1}$) is the volumetric flow rate of the release and $C_s$ (ppm) is the concentration at the source.

## 4. RESULTS

In presenting the results for the cases listed in Table 1, all distances are normalized by the building height $H_b$. Horizontal distances $x, y$ are given in terms of distance from the source position while vertical distances $z$ are given as height above ground.

Figures 2 to 4 present results for the most simple of building configurations, that of a cube at 0° and 45° to the mean wind direction. Numerical and experimental results are shown for releases over the building roof centre at heights $H_b \leq z \leq 1.5H_b$. As can be seen in Figure 2 maximum ground level concentrations (GLC) are predicted very well by the model. The average difference between the numerical and experimental values being 9.3% and 11.9% for the 0 and 45° cases respectively. This is well within the limits of agreement that we might expect between different experimental investigations. It should also be noted again that the vortices for the 45° case were only tuned for the roof top release and then just applied for all other cases.

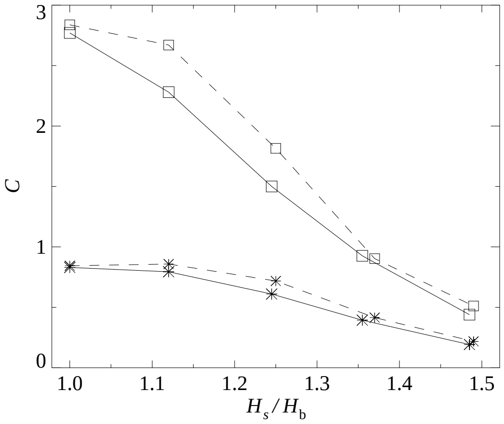

Figure 2: Maximum centre line ground level concentrations $C_{\text{max}}$. Numerical (*dashed line*) and experimental (*solid line*) results for cube; $*$ at 0° and $\square$ at 45°.

However, maximum GLC while perhaps offering vital information as far as safety procedures are concerned is not the most robust means of evaluating real model performance. Figures 3 and 4 provide a more complete impression of model performance. The GLC profile presented in Figure 3 shows for the 0° case that the model performs very well at all downstream distances. The results for the 45° case while still more than acceptable, highlight the lack of information present if only maximum GLC are considered. The early peak and subsequent under prediction of GLC for the 45° case was common for all release heights. Figure 4 shows contours in the $x - z$ plane on the lateral centre plane at $y = 0$. Again we see that the model performs well, capturing the entire structure of the dispersing plume. This realistic representation of the whole building effected plume structure is seen as a significant feature of the current approach that extends the models suitability far beyond simple assessments.

Results for two further building geometries are also presented here. Figure 5 shows maximum GLC for the building $2H_b = W_b = L_b$. Here the 0° experimental results are very similar to those for the cube. However the experimental results for the 45° case are significantly different to those for the cube even though the building aspect ratio change is not large. The model performs well for both orientations, capturing both the aspect ratio and orientation interaction.

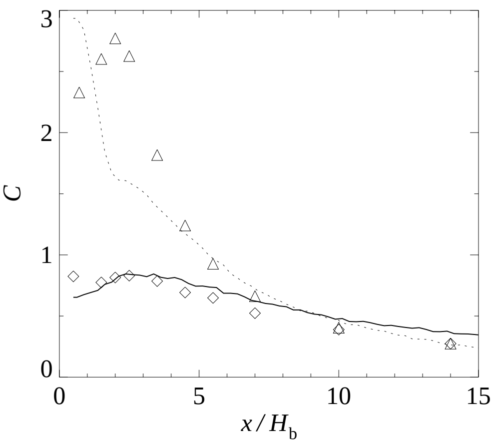

Figure 3: Centre line ground level concentrations $C$. Numerical (*lines*) and experimental (*symbols*) results for a cube at 0° (◇ & *solid line*) and at 45° (△ & *dashed line*).

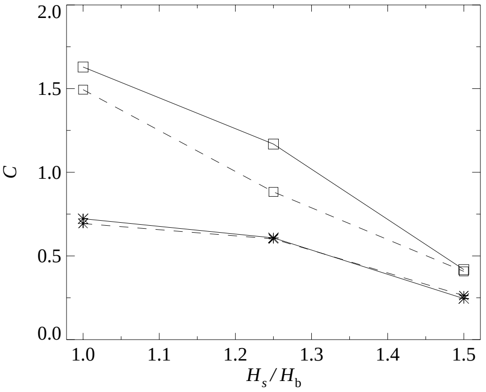

Figure 5: Maximum centre line ground level concentrations $C_{\text{max}}$. Numerical (*dashed line*) and experimental (*solid line*) results for cuboid $2H_b = W_b = L_b$; $*$ at 0° and $\square$ at 45°.

Figure 6 presents results for a building of height to width/length aspect ratio of 3:1 for a height of 120 m. This represents an extreme aspect ratio building but also one that is characteristic of several industrial process facilities. For both building orientations the model performs well in terms of both peak concentration and also in plume distribution. In particular the experimental results for the 45° case show a double peak in the GLC in the near wake. This would appear to be the result of a separation between the effluent entrained into the recirculation region and the rest of the plume acting under the mean stream line curvature. What is very encouraging is that the model has reproduced, to a large extent, this complicated structure. Although it is noted that the model also produces a small recirculation GLC peak for the 0° case which was not seen experimentally. However, it is believed that this highlights the robustness and wide ranging applicability of the current modeling approach.

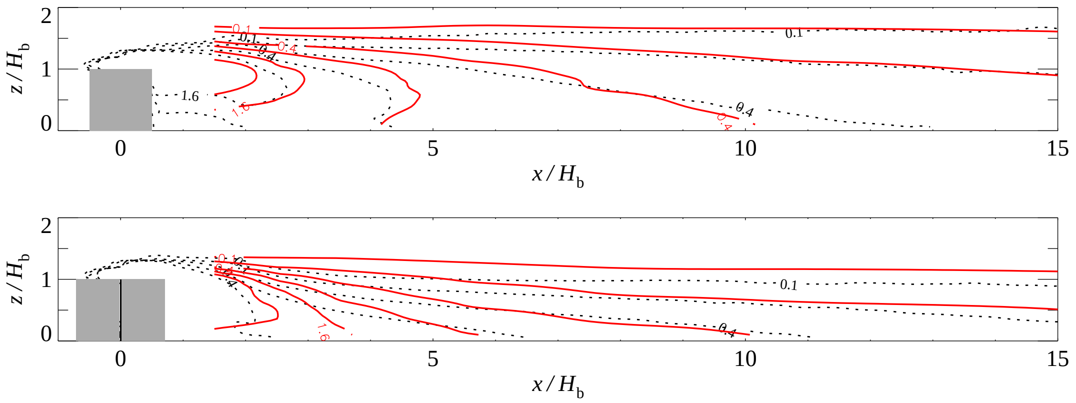

Figure 4: Concentration contours in the $x - z$ plane at $y = 0$ for an isolated cube at 0° (*top*) and 45° (*bottom*). Release at building centre roof top $(x, y, z)/H_b = (0, 0, 1)$.

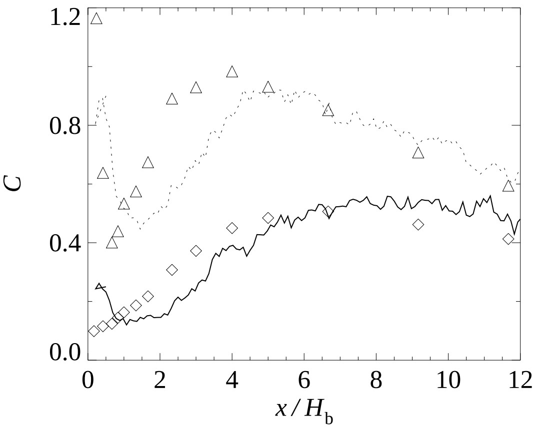

Figure 6: Centre line ground level concentrations $C$. Numerical (*lines*) and experimental (*symbols*) results for a cuboid $H_b = 3W_b = 3L_b$ at 0° (◇ & *solid line*) and at 45° (△ & *dashed line*).

## 5. CONCLUSIONS

A new Lagrangian stochastic model for the prediction of dispersion in the vicinity of cuboid buildings submerged in a deep neutrally stable atmospheric boundary layer has been presented. Comparison with laboratory experimental data has been conducted as part of the model development and evaluation. This included comparison of both ground level and elevated concentration measurements for cuboid buildings of height to width aspect ratios ranging from 3:1 to 1:2, for passive emissions at heights ranging from 1 to 1.5 building heights and for building orientations of 0° and 45° to the mean flow. The model has been shown to capture very well both the magnitude and the three dimensional spatial distribution of the resulting concentration fields.

## References

Counihan, J., Hunt, J. C. R. and Jackson, P. S.: 1972, Wakes behind two-dimensional surface obstacles in turbulent boundary layers, *J. Fluid Mech.* **64**, 529–563.

Fackrell, J. E.: 1984, Parameters characterising dispersion in the near wake of buildings, *J. Wind Eng. and Ind. Aero.* **16**, 97–118.

Hort, M. C. and Robins, R. G.: 2001, The dispersion of fugitive emissions from storage tanks, *Submitted to J. Wind Eng. and Ind. Aero.* .

Li, W. W. and Meroney, R. N.: 1983a, Gas dispersion near a cubical model building. Part I. Mean concentration measurements, *J. Wind Eng. and Ind. Aero.* **12**, 15–33.

Li, W. W. and Meroney, R. N.: 1983b, Gas dispersion near a cubical model building. Part II. Concentration fluctuation measurements, *J. Wind Eng. and Ind. Aero.* **12**, 35–47.

Robins, A. G. and Apsley, D. D.: 2000, ADMS 3 technical specification: Modelling of building effects in ADMS, *Technical Report P16/01N/00*, CERC, Cambridge, UK.

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. Env.* **11**, 291–297.

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. Env.* **11**, 299–311.

Snyder, W. H.: 1994, Downwash of plumes in the vicinity of buildings: A wind tunnel study, *Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes* pp. 343–356.