Wind and Structures, Vol. 5, No. 2~4 (2002) 89-100
DOI: http://dx.doi.org/10.12989/was.2002.5.2_4.089

Flow and dispersion around storage tanks. A comparison between numerical and wind tunnel simulations

C. E. Fothergill† and P. T. Roberts‡

Shell Research Ltd., P.O. Box 1, Cheshire, CH1 3SH, U.K.

A. R. Packwood‡†

School of Mechanical and Materials Engineering, University of Surrey, Guildford, GU2 5XH, U.K.

Abstract. Accidental gaseous losses from industrial processes can pose considerable health and environmental risks but assessing their health, safety and environmental impact is problematic. Improved understanding and simulation of the dispersion of emissions in the vicinity of storage tanks is required. The present study aims to assess the capability of the turbulence closures and meshing alternatives in a commercially available CFD code for predicting dispersion in the vicinity of cubes and circular cylindrical storage tanks. The performance of the k-ε and Reynolds Stress turbulence models and meshing alternatives for these cases are compared to experimental data. The CFD simulations are very good qualitatively and, in many cases, quantitatively. A mesh with prismatic elements is more accurate than a tetrahedral mesh. Overall the Reynolds stress model performs slightly better than the k-ε model.

Key words: CFD; near-field dispersion modelling; circular cylinders; storage tanks.

1. Introduction

Accidental gaseous losses from industrial processes pose considerable health and environmental risks but assessing their impact is problematic. On a typical petrochemical process site approximately 50% of total site losses arise from the tanks in which feedstock and products are stored. This occurs during filling, by evaporation, and from the failure of seals. Of particular health and safety concern are the concentrations of gas in the vicinity of tanks. Current methods of assessing gas concentrations in these areas involve extensive measurement or the use of models which are unable to account directly for the effects of complicated process plant structures on flow and dispersion. Therefore improved understanding and modelling of the dispersion of emissions in the vicinity of storage tanks is required. Many storage tanks are surrounded by a low-lying surrounding wall known as a bund. These are used on industrial sites to capture the contents of the storage tank if it fails. Their effect on flow and dispersion is of interest in addition to providing a challenging test case for CFD.

Numerous experimental studies have investigated flow, and in some cases, dispersion, around

† Research Engineer
‡ Senior Consultant
‡† Lecturer

circular cylinders (Probert 1973). Holroyd (1983) reported flow structures around short surface mounted cylindrical obstacles submerged in a deep boundary layer, in particular the horse-shoe vortex with vertical dimension the same order as the building height. Lim and Lewkowicz (1986) found a recirculating zone to extend approximately one building height from the downwind end of the obstacle.

Computational studies of flow and dispersion around buildings are not as numerous as experimental studies. Lakehal (1999) computed the flow around circular cylinders of finite length and used surface roughness to simulate the effect of higher turbulence on the separation point. Two separate studies of flow inside a floating-roof storage tank using CFX-4 have been found to agree well with full-scale measurements (Fothergill 1998, Pasley 2000). Meroney (1999) investigated the flow and dispersion of gases near different building shapes. Concentrations were consistently over-predicted in the vicinity of bluff bodies by numerical models using Reynolds-averaged turbulence models, although the Reynolds Stress model produced somewhat more realistic results than the k-ε or RNG models.

Numerical studies of dispersion around finite length circular cylinders is an area of interest that has not been subsequently addressed. The present study aims to assess the capability of the turbulence closures and meshing capabilities in a commercially available CFD code for predicting dispersion in the vicinity of cylindrical storage tanks. The reliability of CFD solutions for turbulent flow is strongly influenced by the turbulence model, particularly in complex flows around buildings and structures. The original k-ε model proposed by Launder and Spalding (1974) is based on the eddy-viscosity concept. It assumes that the turbulent viscosity μ_t is isotropic, in other words that the ratio between Reynolds stress and mean rate of deformation is the same in all directions. This assumption fails in flows which are dominated by strong anisotropic and nonequilibrium effects (Wright and Easom 1999). The Reynolds stress equation model (Launder 1975) provides the extra turbulent momentum fluxes from the solution of full transport equations. This method is more computationally demanding but it accounts for anisotropy and the transport of the Reynolds Stresses. The results of the k-ε and Reynolds Stress turbulence models and meshing alternatives are compared to detailed experimental data.

2. Methodology

Simulations were made of dispersion of a neutral density passive release from the centre top face of four ‘buildings’: a cube at 0° to the flow, a cube at 45° to the flow, and circular cylinders with and without bunds. In every case the ‘building’ height h was 0.1 m and the height to width ratio was 1:1. The buildings were mounted on a rough wall immersed in a boundary layer $\delta/h = 10$, δ being the depth of the boundary layer. The Reynolds number of the flow was 1.54×10^5.

The commercially available CFD code, CFX-5.3, was used for the CFD simulations. This is a finite volume unstructured grid code with a coupled solver. The simulations were run on a Pentium III 800 MHz using the Microsoft NT operating system.

The transport equation for a scalar in the presence of turbulence is given by

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (U\phi) - \nabla \cdot \left(\left(\rho D_{\phi} + \frac{\mu_t}{S_{c_t}} \right) \nabla \cdot \left(\frac{\phi}{\rho} \right) \right) = S_{\phi} \tag{1}$$

where ρ is the density, ϕ is the concentration, ϕ/ρ is the conserved quantity per unit mass, S_{ϕ} is a

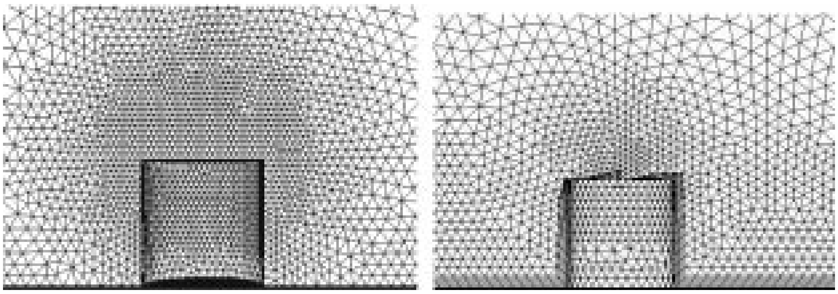


Fig. 1 Unstructured mesh of tetrahedral elements and tetrahedral mesh with a layer of 5 prismatic elements on walls

volumetric source term, D_ϕ is the kinematic diffusivity of the scalar and μ_t is the turbulent viscosity, with S_{ct} the turbulence Schmidt number.

Velocity, turbulence and concentration were measured in the Enflo atmospheric boundary layer wind tunnel at the University of Surrey (Hort and Robins 1999). CUL^2/QC_s from the wind tunnel and simulations were compared, where C is measured concentration, U the reference velocity, L the reference length scale, Q the volumetric flow rate of the release and C_s the concentration at the source.

A solution domain extending 5h vertically, laterally, and upwind of the obstruction, and 15h downwind of the obstruction was used, following the guidelines in the Evaluation of Modelling Uncertainty report (Hall 1996).

The effect of grid cell shape was investigated. In near wall regions gradients are greatest normal to the surface. Computationally efficient meshes in these regions require that the elements are finely resolved normal to the wall, but coarse parallel to it. This requirement is addressed by using a layer of prismatic, 5-sided elements near all walls (Fig. 1). Solutions on completely tetrahedral meshes, and meshes with a layer of prismatic elements near the surfaces, were compared. It was ensured that the solution was independent of grid cell size in the region of the body.

The inlet boundary conditions were matched to the wind tunnel measurements of velocity and turbulence parameters. A hybrid differencing scheme (Spalding 1972) was used for all simulations.

3. Results and discussion

The mesh with prismatic elements on the walls is superior to the tetrahedral mesh in a number of ways. It predicts the length to reattachment of the recirculation region behind the cube at 0^o to the flow and the cylinder (Table 1), and the flow and turbulent kinetic energy (Figs. 2 and 3) more accurately. The drop of the plume to ground level behind the cylinder found in the wind tunnel is only captured on the mesh with prisms (Fig. 7), due to the prediction of the recirculation region behind the cylinder (Fig. 2) which carries the plume to the ground. The mesh with prisms also captures the lateral spreading better than the tetrahedral mesh (Figs. 9 and 10). The mesh with prisms on the walls gives improved predictions in the vicinity of curved wall geometries because prediction of the point of flow separation from curved walls requires the turbulent energy normal to the wall to be achieved precisely.

The fine mesh resolution normal to the walls models these rapid gradients and is closer to

Table 1 Comparison of distance to reattachment of recirculation region measured from centrepoint of obstruction. All simulations and experiments have similar Reynolds numbers

	Length to reattachment, x / h	
Turbulence model and mesh type	Cube at 0° to the flow	Cylinder
k-ε model, tetrahedral mesh	2.35	1.35
k-ε model, mesh with prisms	2.3	1.55
RS model, tetrahedral mesh	2.6	1.2
RS model, mesh with prisms	2.3	1.75
Experiment (Meroney; Robins)	2.28	1.6

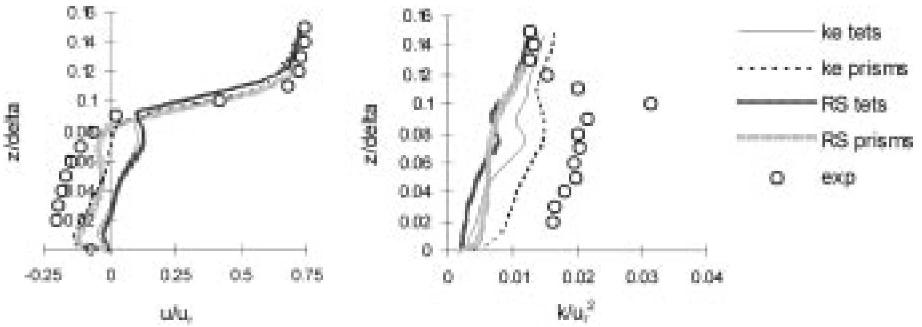


Fig. 2 u velocity and turbulent kinetic energy, (u_r is u at $z = \delta$), 3/4h downwind of cylinder centreline

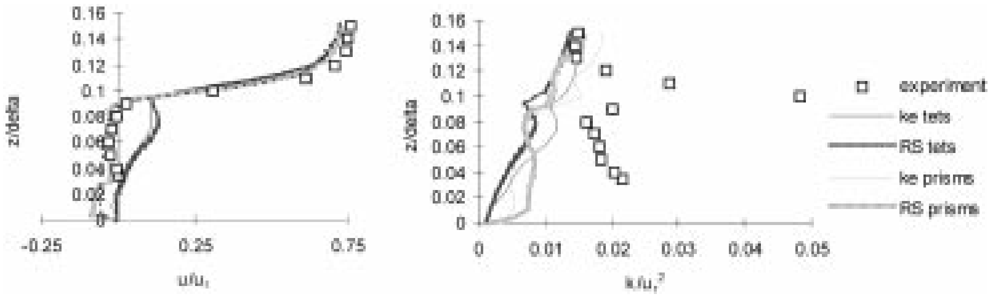


Fig. 3 u velocity and turbulent kinetic energy, 3/4h downwind of cylinder with bund on centreline

achieving the correct point of separation from the body. This is key in determining the extent of the recirculation region and the plume characteristics. The point of separation from a sharp edged geometry is easier to predict, so separation, recirculation and concentrations behind the cubes is predicted similarly well with either mesh.

One distinct difference between the k-ε and Reynolds stress models is the higher prediction of turbulent kinetic energy given by the k-ε model on the windward side of the obstructions shown in Fig. 4. This is due to the difference in the modelling of the turbulence production term P_k. Eddy viscosity models tend to return excessive levels of energy and turbulent diffusion in the

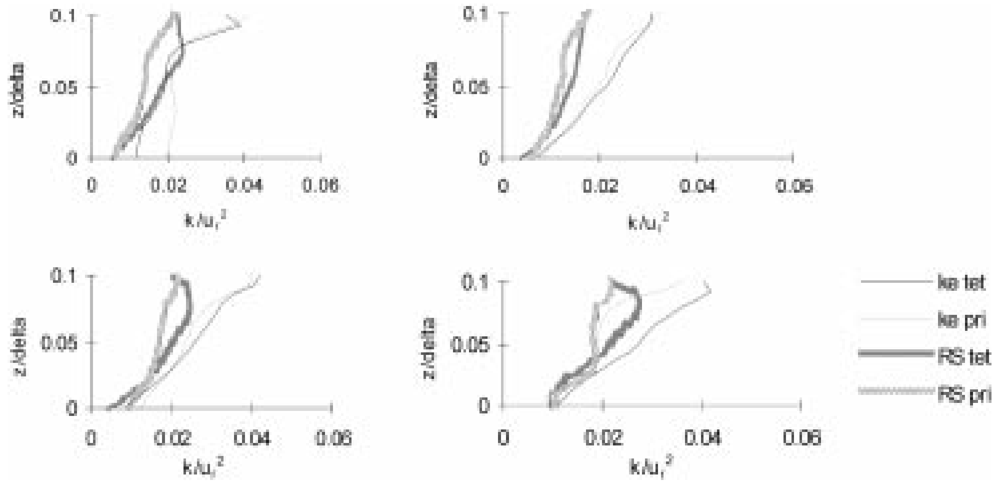


Fig. 4 Turbulent kinetic energy on the windward side of the obstruction. From top left, cube at 0° to the flow, cube at 45° to the flow, cylinder and cylinder with bund

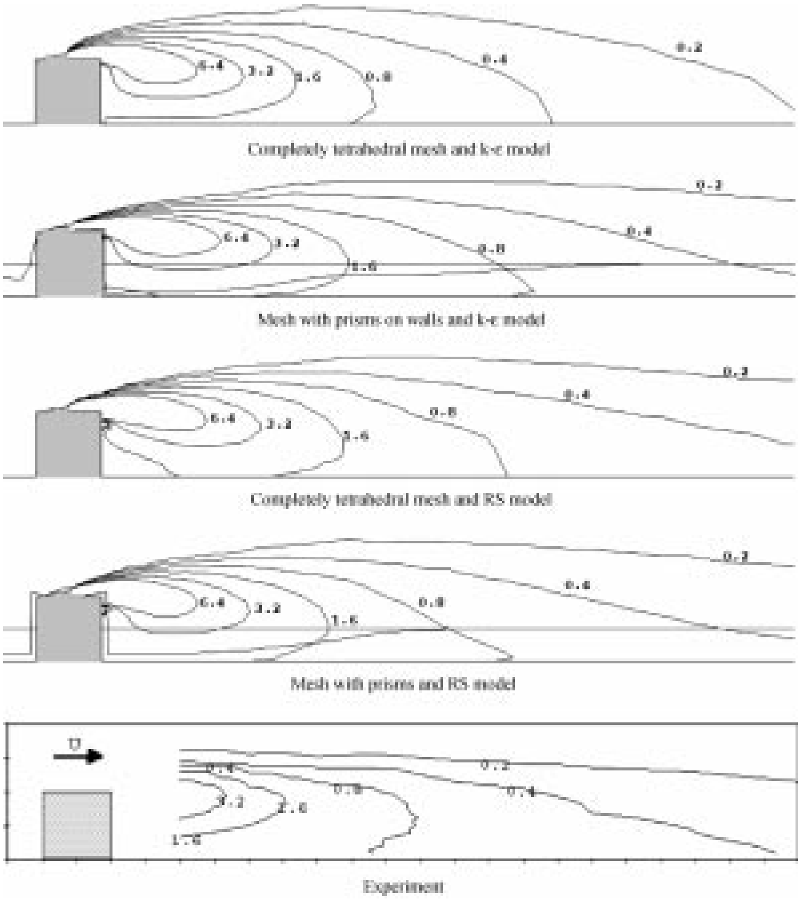


Fig. 5 CUL^2 / QCs in centreline downwind of cube at 0° to the flow

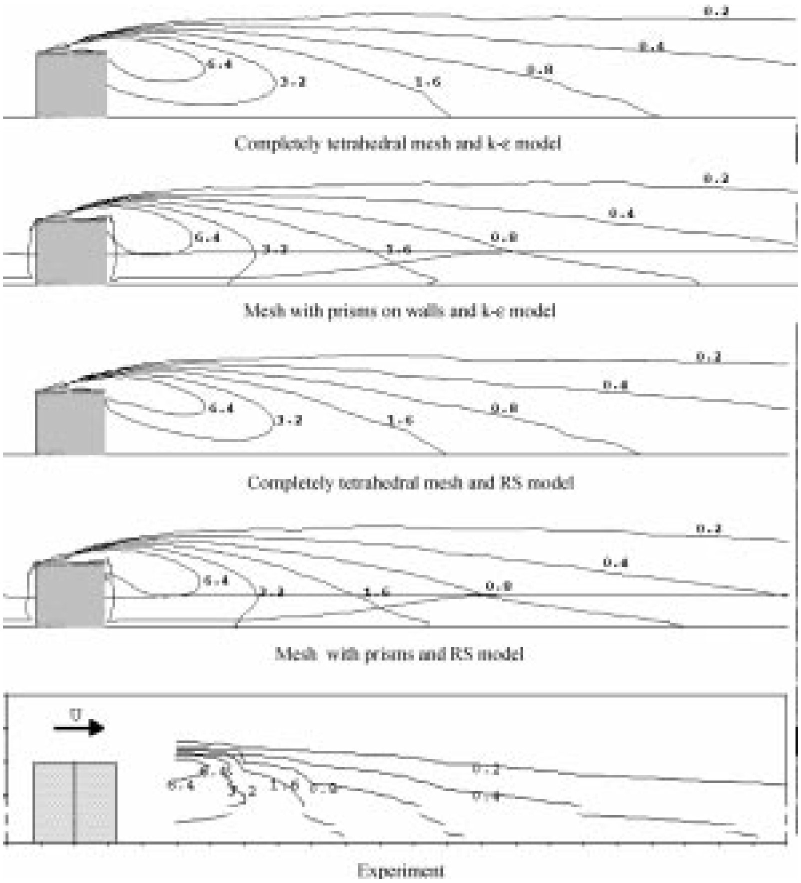


Fig. 6 Centrelind CUL^2 / QC_s in the wake of a cube at 45° th the flow

presence of strong compressive strain. One possible manifestation of this is late separation and a narrower wake, resulting in less lateral spreading which is predicted behind the cylinders and the cube at 0° to the flow by the *k*-*ε* model (Figs. 9 and 10).

The prediction of turbulence, flow and concentration in the wake are otherwise quite similar from both turbulence models. In most cases the alongwind concentrations, and in every single case the vertical concentrations, are too high by between 50-100% at more than 1.5*h* downwind (Figs. 5-10). This may be due to the fact that turbulent kinetic energy (Figs. 2 and 3) and hence turbulent diffusion in the wake is too low. The wake region exhibits strong turbulence anisotropy where the *v*'*v*' lateral Reynolds stress component dominates (Murakami 1993). The *k*-*ε* model underestimates the value of *v*'*v*' in the wake region.

The net result is that the kinetic energy, calculated as the sum of the Reynolds stress, is underestimated by the *k*-*ε* model. Lateral spreading at 10 *h* downwind is very accurate behind the cube 0° and bunded cylinders but is underpredicted behind the cube at 45° case and the cylinder case (Figs. 9 and 10). The Reynolds stress model can predict anisotropic turbulence so it was expected that it would capture the preferential lateral spreading and reduced vertical spreading caused by the domination of the lateral Reynolds stress component (Hall 1997). The *k*-*ε* model

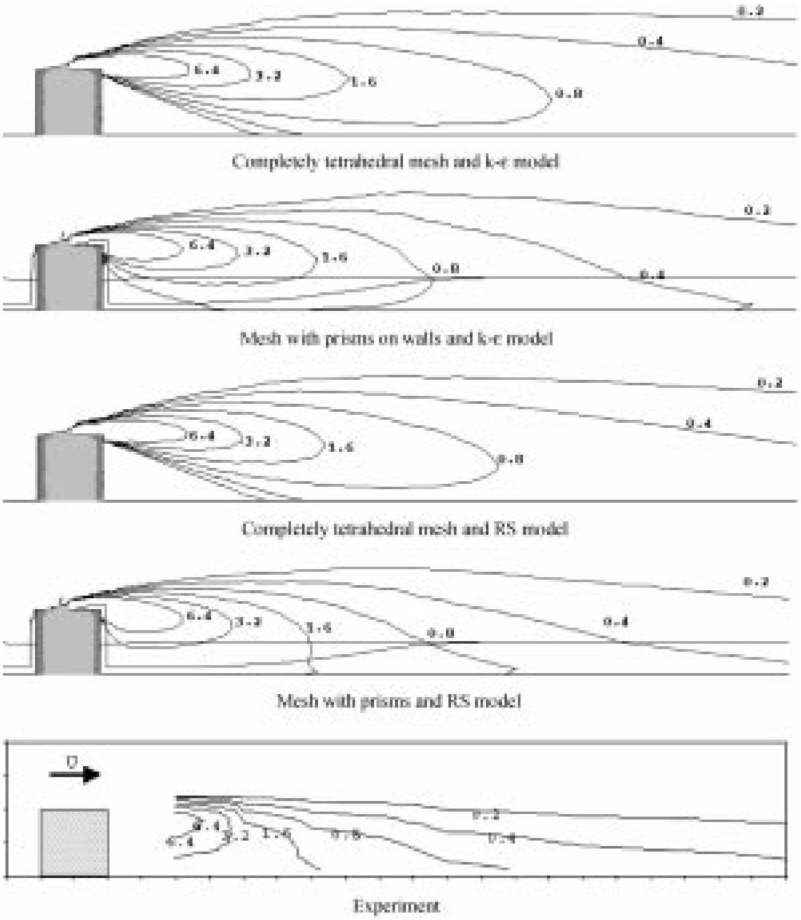
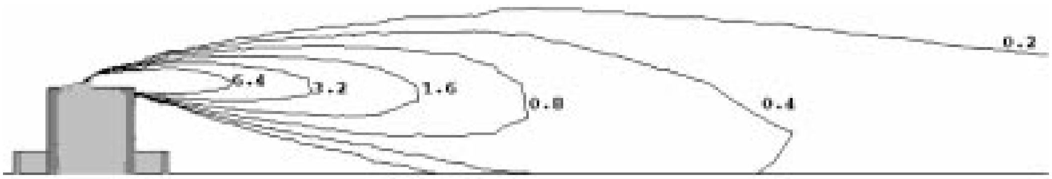
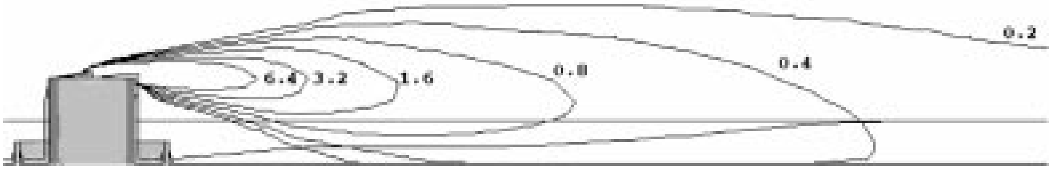


Fig. 7 Centreline CUL^2 / QC_s in the wake of a cylinder

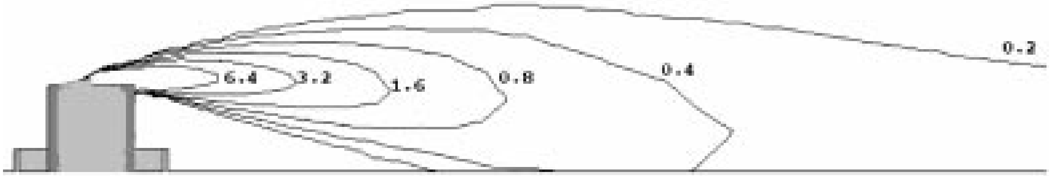
provides an isotropic treatment and therefore tends to give similar spreading rates in the three co-ordinate directions. However the Reynolds stress model predicted the Reynolds stresses to be near isotropic. This may be due to the isotropic distribution of the Reynolds stresses at the inlet. Solutions were tested for grid sensitivity at a profile 1.5h downwind of the obstructions so the possibility that the solution is grid sensitive at greater than 1.5h downwind remains. The k-ε and RS models were nonetheless run using the same mesh and should be comparable. The new release of CFX-5 will incorporate adaptive meshing which will reduce uncertainty due to numerical errors and grid-sensitivity. Uncertainties in wall modelling need to be addressed. The wall functions used in the k-ε model do not differentiate between the effects of viscosity and wall proximity on the turbulent motion, leading to poor predictions of flow regions where these effects take different magnitudes. In the Reynolds stress model the pressure-strain correlation uses the 'isotropization of production' model of Launder, Reece and Rodi (1975), which does not account for wall reflection, and performs poorly in swirling flows and impingement regions (Craft and Launder 1991).



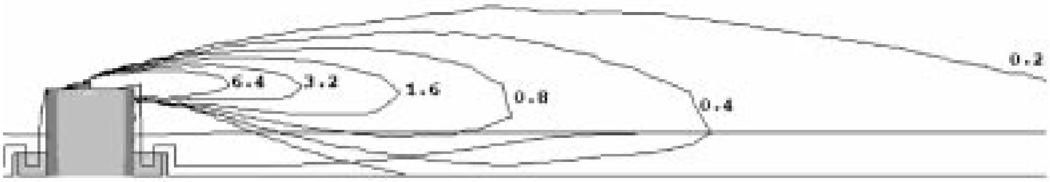
Completely tetrahedral mesh and k-ε model



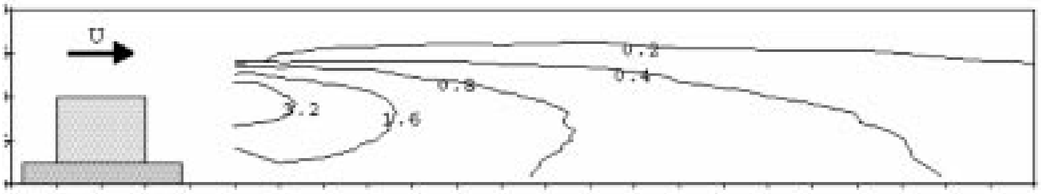
Mesh with prisms on walls and k-ε model



Completely tetrahedral mesh and RS model



Mesh with prisms and RS model



Experiment

Fig. 8 Centreline CUL^2 / QC_s in the wake of a cylinder with a bund

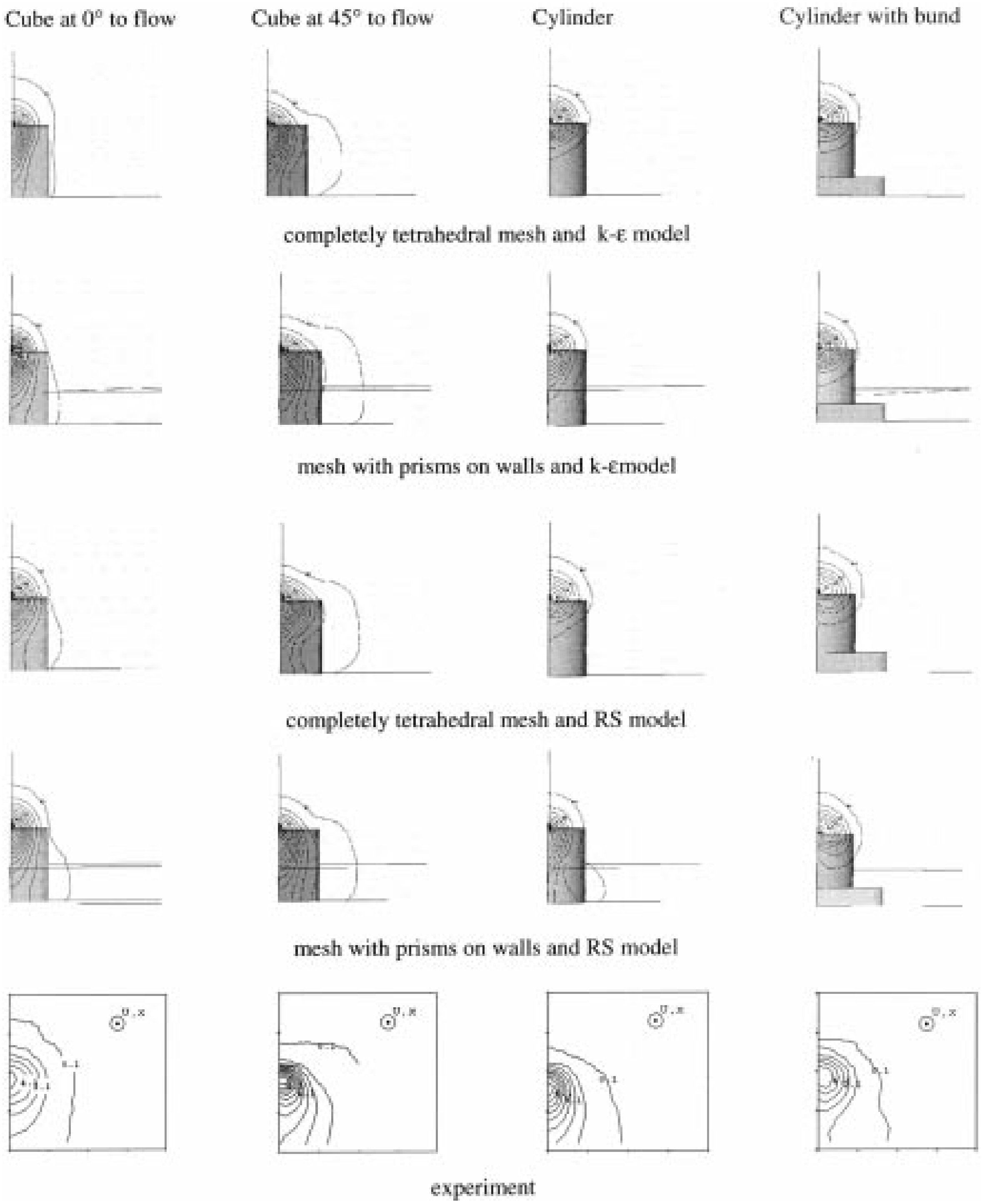


Fig. 9 CUL^2/QC_s at 15h downwind of obstruction centrepoint

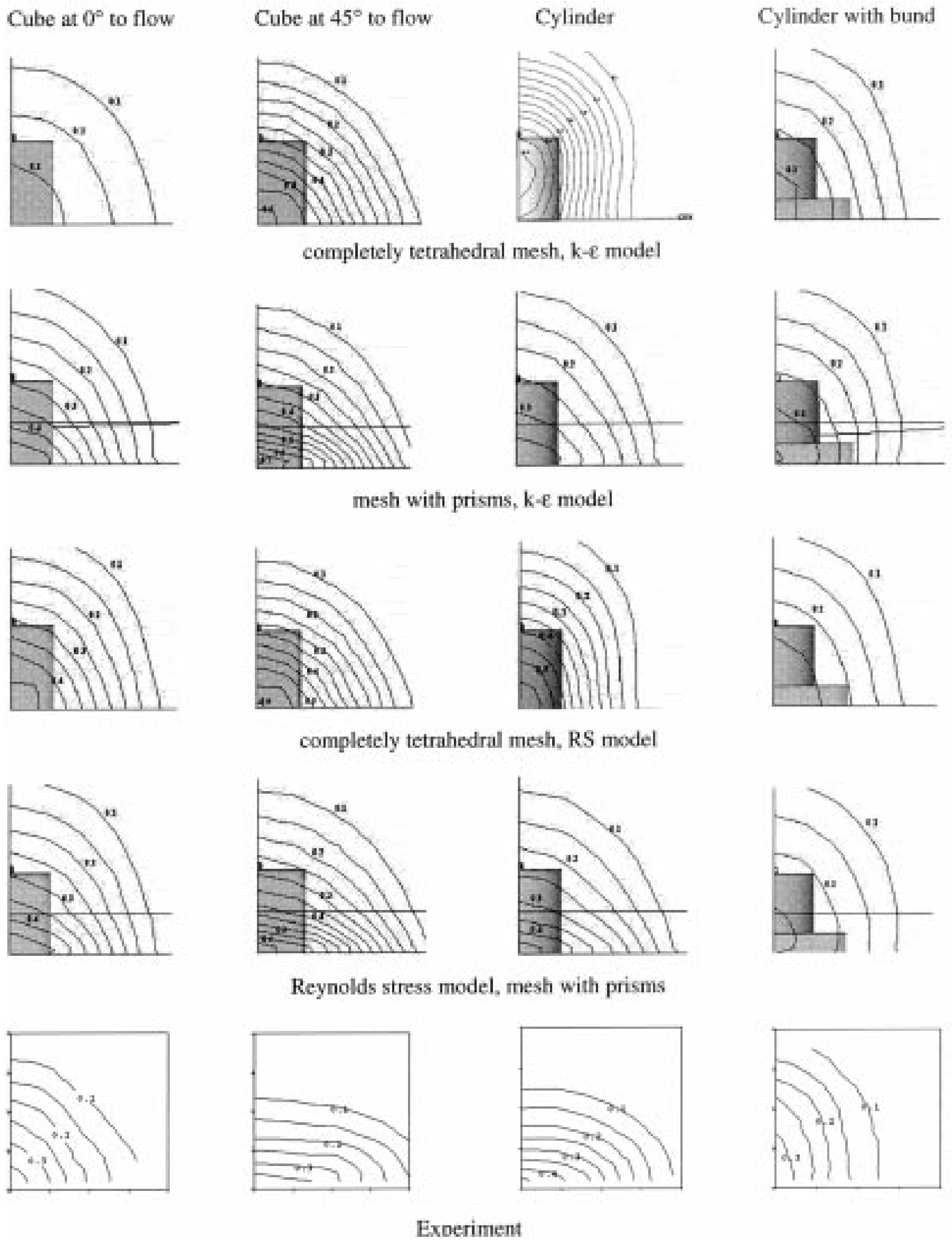


Fig. 10 CUL^2/QC_s at 10.5h downwind of obstruction centrepoint

4. Conclusions

The study shows that solutions are more dependent on the details of mesh design than the turbulence model. Meshes with prismatic elements on the walls predict flow, turbulence, and qualitative characteristics of dispersion more accurately, especially for curved wall geometries. The higher prediction of turbulence at the leading edge of the obstructions by the k-ε model may lead to later separation, a smaller recirculation and a narrower wake. The dispersion predictions from the two turbulence models hardly differs, which may be due to the isotropic distribution of the Reynolds stresses at the inlet of the Reynolds stress model cases. Turbulent kinetic energy is generally too low in the wake. Concentrations are too high but within a factor of two of the measurements. Uncertainties in the treatment of boundary conditions, the sensitivity of the solution to the grid, and numerical errors may be obscuring the real performance of the turbulence models, all of which highlights the continued difficulty for CFD practitioners in obtaining reliable solutions.

Despite the shortcomings of the model, the discrepancies in solutions are less significant than those obtained from simpler models. CFD can provide resolution of concentration gradients in regions of separated and secondary flows, without making the gross assumption of uniform concentration that is sometimes used in simple wake cavity and street canyon parameterisations. The present study has therefore enhanced understanding of flow and dispersion characteristics in the vicinity of storage tanks which is useful in guiding the use of alternative models such as ADMS. The detailed information provided by CFD simulations can improve the efficiency of the calculation of emission rates and help identify sources of fugitive emissions. Overall the study has increased understanding of the errors and variability involved in the application of numerical techniques, but further work is required to judge their use from an informed position.

Acknowledgements

The author would like to thank Dr. Martin Tasker, Eutech, for sponsoring the experimental investigations and providing unpublished results, and Prof. Alan Robins and Dr. Matthew Hort, University of Surrey, for carrying out the experimental investigations. The numerical simulations are funded by the Engineering and Physical Sciences Research Council and Shell Research Ltd.

References

Craft, T.J. and Launder, B.E. (1991), "Computation of impinging flows using second-moment closures", *Eighth Symp. Turb. Shear Flows*, Munich, 8-5.

Fothergill, C.E. "Flow over a floating roof tank. Comparison of field data and CFD simulations", *Shell RTS Report* OP.99.47120.

Hall, R.C., et al. (1996), "Evaluation of modelling uncertainty: CFD modelling of near-field atmospheric dispersion", *European Commission Contract EV5V-CT94-0351*.

Hall, R. (1997), "Dispersion of releases of hazardous materials in the vicinity of buildings. Phase II - CFD modelling", London, HSE Research Contract RSU 8000, Report WSA/8013.

Holroyd, R.J. (1983), "On the behaviour of open-topped oil storage tanks in high winds. Part I. Aerodynamic aspects", *J. Wind Eng. Ind. Aerod.*, **12**, 329-352.

Hort, M., and Robins, A. (1999), "The dispersion of fugitive emissions from storage tanks: ground level concentration measurements", Unpublished, Report completed for Huntsman, University of Surrey ME-FD/ 99.102.

Lakehal, D. (1999), "Computation of turbulent shear flows over rough-walled circular cylinders", *J. Wind Eng.*

Ind. Aerod., **80**, 47-68.

Launder, B.E., and Spalding, D.B. (1974), "The numerical computation of turbulent flows", *Comp. Meth. App. Mech. Eng.,* **3**, 269-289.

Launder, B.E., Reece, G.J. and Rodi, W. (1975), "Progress in the development of a Re-stress turbulence closure", *J. Fluid Mech.,* **68**, 537-566.

Lim, F.K. and Lewkowicz, A.K. (1986), "Investigation by laser doppler anemometry of the turbulent flow around cylindrical obstacles on a rough surface", *Int. J. Heat & Fluid Flow,* **7**(2), 102-108.

Meroney, R.N. (1999), "Wind-tunnel and numerical modelling of flow and dispersion about several building shapes", *J. Wind Eng. Ind. Aerod.,* **81**, 333-345.

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

Pasley, H. and Clark, C. (2000), "Computational fluid dynamics study of flow around floating roof storage tanks", *J. Wind Eng. Ind. Aerod.,* **86**, 37-54.

Spalding, D.B. (1972), "A novel finite-difference formulation for differential expressions involving both first and second derivatives", *Int. J. Numer. Methods Eng*., **4**, 551-559.

Wright, N.G. and Easom, G.J. (1999), "Improved computational models for wind engineering", *10th Int. Conf. on Wind Eng.,* Copenhagen June 1999.