AIVC 12035

Modelling Three Dimensional Gravity-Induced Natural Convection Buoyant Plumes

J. Häkkinen, G. Stewart, P. Sarkomaa

Dept. of Energy Technology
Lappeenranta University of Technology
Lappeenranta, FINLAND

ABSTRACT

The aim of this study was to ascertain the validity of using computational fluid dynamics (CFD) techniques to predict the behaviour of three dimensional gravity-induced natural convection buoyant plumes from a vertical heated cylinder in a large quiescent enclosure.

The calculated velocity distributions and turbulence quantities over the cylinder were compared to a wide range of experimental measurements. The laminar boundary layer on an isothermal vertical plate was also modelled.

The CFX4.1 code with the CCCT (QUICK-based) discretisation method and low Reynolds number $k\varepsilon$-turbulence model was used. For buoyancy, the Boussinesq approximation was employed. In the calculations, three different sets of boundary conditions were studied.

The CFD calculations satisfactorily predicted the boundary layer, velocity profiles, magnitudes and spreading rates of the plume. Good agreement between the CFD predictions for the turbulence intensities and the experimental data was also achieved.

KEYWORDS

CFD, Modelling, Natural convection, Plumes

INTRODUCTION

Gravity-induced buoyant plumes in quiescent surroundings have many industrial applications, such as the heating of room by a convector heater or the cooling of electronic components. In industrial processes like welding, for instance pollutants or other undesirable gases are released in the form of thermal plumes. Hence knowledge of their behaviour is important for efficient ventilation and air conditioning.

There are two ways to study buoyant plumes, namely, by experimental or by computational fluid dynamics (CFD) techniques. If CFD is found to be reliable, costly experiments can be replaced with more simple and economical CFD methods. Nevertheless, at this stage one should be cautious when interpreting CFD results and their validation by experimental measurements is strongly advised.

The objective of this study is to model the turbulent three-dimensional buoyant plume from a vertical cylinder heat source using CFX 4.1 code and use the wide range of available experimental data for validation purposes. The experimental data also contains the fluctuation terms for the velocity components. This information is not of particularly great interest as far as the study of the flow conditions is concerned, here the velocity profile is the most important parameter. However, it is important in that it may be used as an indication of the accuracy of the use of computational fluid dynamics techniques as a solution method for this kind of natural convection problem.

A secondary objective is to evaluate the suitability of CFD techniques for modelling natural convection laminar boundary layer velocity and temperature profiles. This may be achieved by comparing these predictions with those obtained using a theoretical approach.

METHODS

CFX4.1 code was used with the CCCT-discretisation method, low Reynolds number kε turbulcnce model, and Boussinesq approximation for buoyancy. The CCCT scheme is based on the upwind QUICK-scheme (Quadratic Upwind Differencing), but it is bounded to prevent non-physical overshoots such as a negative turbulence kinetic energy. The low Reynolds number kε-model is modified from the standard kε-model to take into account the wall effects of surfaces, and flows undergoing laminar-turbulent transition. This model involves a damping of the eddy viscosity when the local turbulent Reynolds number is low, a modified definition of ε so that it goes to zero at walls, and modifications of the source terms in the ε equation. The low Reynolds number kε turbulence model is valid for flows with a Reynolds number of up to 30000. The Boussinesq approximation was used for buoyancy in these calculations. In the Boussinesq approximation, the density is assumed constant except in the buoyancy terms in the momentum equation. Also, an extra source term for turbulence kinetic energy is added to the low Reynolds number kε turbulence model. The Boussinesq approximation is recommended for cases involving small temperature differences, such as is in this study.

Laminar natural convection boundary layer

The suitability of using CFX-code to predict laminar free convection velocity and temperature profiles on a vertical surface in a quiescent environment was investigated. The boundary layer development on a flat plate was calculated for a flow not exceeding the critical Rayleigh number of 10^9 (see Figure 1).

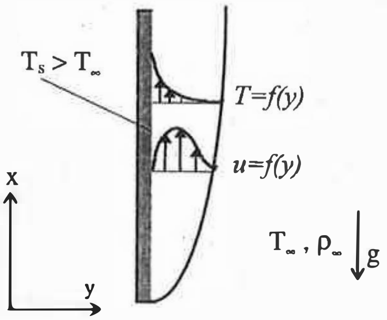

Figure 1 Laminar free convection velocity boundary layer on an isothermal vertical surface.

Computational domain

The heat source used was a vertical cylinder of diameter 0.32 m and height1.2 m. The laboratory, which had a volume of 728 m^3, was considered as a quiescent space. The computational domain was chosen so as to maintain a similar height and volume to that of the laboratory in which the experiments were conducted. A schematic diagram of the computational domain is illustrated in Figure 2. The generated computational grid contained some 27000 cells, and is shown in Figure 3.

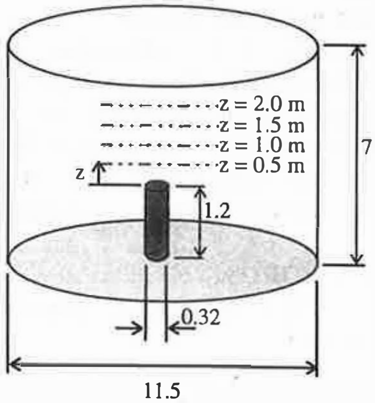

Figure 2 Schematic diagram of computational domain

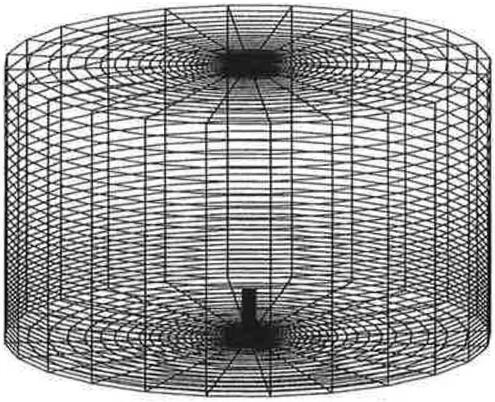

Figure 3 The computational grid

Boundary conditions

For the boundary conditions, a constant heat flux of 425 W/m^2 was set for the cylinder surface. This corresponds to the convection heat transfer rate of the experiments by Welling 1993. For the outer boundaries, three sets of boundary conditions were studied: reference pressure at the top, walls all around, and walls all around that the global energy balance was satisfied, i.e., the heat released from the cylinder was equal to that lost from the outer walls.

Validation data

The CFD results were validated by comparing them to the measured data of Welling (1993). The experimental set-up consisted of a vertical cylinder as the heat source which was placed directly on the floor of a large laboratory of height at least 7 m and volume at least 728 m^2. The concrete cylinder was 1.2 m in height, 0.32 m in diameter and was painted black. The heat source contained internal electrical heating coils which dissipated 1200 W. It was estimated that the total convection heat transfer rate from the surface of the cylinder was approximately 460 W. There was no mechanical ventilation used during the experiments and transparent plastic walls (3 × 3.5 m) were positioned around the cylinder in an effort to minimise disturbances from the surroundings.

Velocity measurements, which were performed using a laser Doppler anemometer (LDA) and a Kaijo Denki 3-D ultrasonic anemometer (KA), were used to yield velocity distributions and fluctuation components in the buoyant plumes. The accuracy of these instruments was estimated as being within a maximum error range of ± 2 %.

With the LDA, vertical velocities were measured at 0.1 m intervals along one radial axis of the cylinder from -1.0 m to 1.0 m (with the origin at the axis of the cylinder) at heights of 0.5, 1.0, 1.5 and 2.0 m above the top of the cylinder. The KA measurements were taken on horizontal planes 0.5, 1.0 and 1.5 m above the top of the cylinder. The origin of the plane was the axis of the cylinder and the range was from -1.0 m to 1.0 m in both directions, with 0.1 m intervals between measurement positions. The measured velocity components were fitted with a Gaussian distribution. Considerable wandering of the plume was observed when the experiments were being conducted. This may be attributed to the turbulent nature of the plume and to disturbances in the surrounding environment. The randomly fluctuating nature of the plume and the finite time during and between data collections at the various measurement positions should be kept in mind when interpreting the results.

RESULTS

The CFD calculations satisfactorily predict the boundary layer, velocity profiles, magnitudes and spreading rates of the plume. Good agreement between the CFD predictions for the turbulence intensities and the experimental data was also achieved.

For the laminar free convection boundary layer, the CFD prediction was in excellent agreement with the theoretical solution of Ostrach (1953) as presented in Schlichting (1979). Both the predicted velocity profile and the temperature profile match almost exactly with the theoretical values.

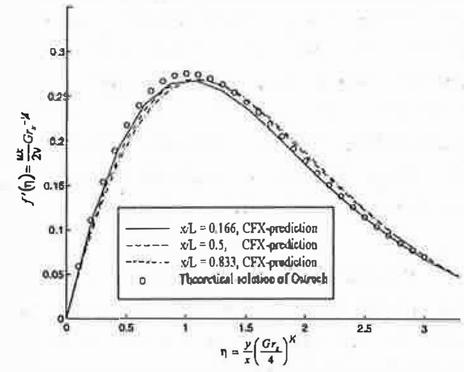

Figure 4 Dimensionless velocity profile of laminar free convection boundary layer on a vertical isothermal surface (Gr=1.8×10^8,Ra= 1.3×10^8).

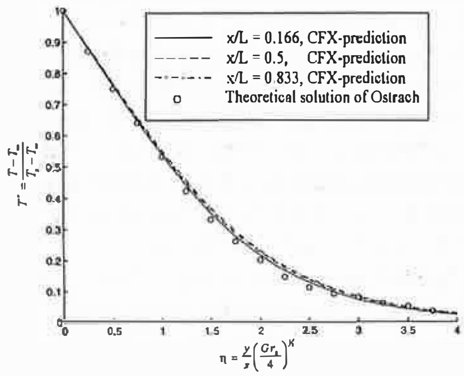

Figure 5 Dimensionless temperature profile of laminar free convection boundary layer on a vertical isothermal surface (Gr=1.8×10^8, Ra = 1.3×10^8).

The three-dimensional turbulent buoyant plume was calculated with three different boundary conditions on the outer boundaries of the domain: reference pressure, walls, and walls maintaining the global energy balance. For the case of the pressure boundary, the ceiling of the laboratory was not included, even though it is known to have an effect on flow field. For the case of having the surrounding walls with a negative heat flux, the vertical velocity profiles became unrealistically flat. These boundary conditions also prevented the wandering of the plume, that was seen during the experiments. For non-conducting walls all around the computational domain, the absolute conservation of energy is not valid, even if the heat transfer rates are relatively small. If this was the case, the calculations became transient and the wandering of the plume could be seen. The results displayed below are for this case of having non-conducting walls all around.

The calculated velocity profiles agree satisfactorily with the experiments, as can be seen from Figure 6.

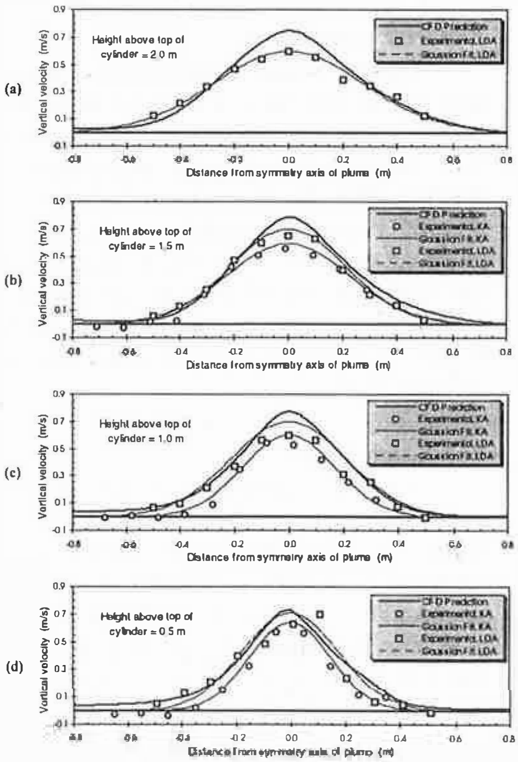

Figure 6 Comparison of CFD predicted vertical velocity profiles and experimental data for various heights above the heated cylinder.

The magnitudes of the vertical velocity are slightly overpredicted. This might be due to the approximated convective heat flux of 425 W/m^2 that is underestimated. However predicted profile and the spreading rate seem to be in a good agreement with the experimental data.

The magnitudes of the turbulence kinetic energy from the CFD calculations are quite close to those from the experiments as can be seen from Figure 7. The typical profile characteristics (as documented, for example by George, Alpert and Tamanini 1977 in the experimental study) can also be seen.

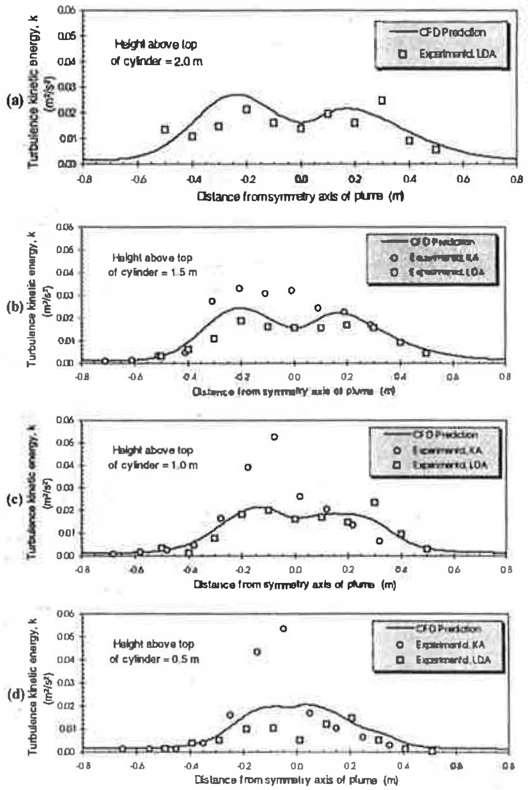

Figure 7 Comparison of CFD predicted turbulence kinetic energy profiles and experimental data for various heights above the heated cylinder.

Two-equation models, such as the low Reynolds number kε turbulence model, assume turbulence to be isotropic. This assumption is deficient when it comes to buoyant plumes as the fluctuations in the radial and circumferential are different to those in the axial directions (Malin and Younis 1990). Nevertheless turbulence must be studied and the two-equation turbulence model used gives a rather good description of the fluctuating behaviour of turbulent flow.

DISCUSSION

The CFD techniques quite reliably predicted the flow field and turbulence in the buoyant plumes. Setting physically and computationally reasonable boundary conditions may be problematic where clear flow boundaries do not exist. The convergence of the solution was found to very sensitive to the changes in the boundary conditions and grid. Hence, a great deal of attention should be paid to the validation of the results.

The magnitudes of the vertical velocities were a slightly overpredicted. This may be partly due to the definition of the convection heat flux from the cylinder where the value of 425 W/m^2 was approximated.

Though the turbulence quantities, such as turbulence kinetic energy, for instance, are in good agreement with the experiments, the isotropic assumption of the two-equation turbulence models is defective. The magnitude was correctly estimated, but the fluctuations in the radial and circumferential directions are different to those in the in axial direction.

This study is a part of an ongoing research programme concerning buoyant flows. Much work remains to be done before an accurate and comprehensive description of these flow fields may be obtained.

ACKNOWLEDGEMENTS

The authors would like to thank Dr. Irma Welling at Lappeenranta Regional Institute of Occupational Health for contributing the wide range of experimental data for our use.

REFERENCES

CFX 4 User Manual (1994), Oxfordshire, United Kingdom.

Gebhart B., Jaluria Y., Mahajan R. L. and Sammakia B. (1988) *Buoyancy-Induced Flows and Transport*, Hemisphere Publishing Corporation.

George, W. K., Alpert, R. L. and Tamanini F. (1977) *Turbulence Measurements in an Axisymmetric Buoyant Plume*, Int. J. Heat and Mass Transfer, 20, 1145-1154.

Malin, M. R. and Younis, B. A. (1990) *Calculation of Turbulent Buoyant Plumes With a Reynolds Stress and Heat Flux Transport Closure*, Int. J. Heat and Mass Transfer, 33, 2247-2264.

Schlichting, H. *Boundary-Layer Theory*, 7th ed. (1979), McGraw-Hill.

Welling, I. (1993) *An investigation of the SVD-Method for Analyzing Convection Plumes*, Lappeenranta Regional Institute of Occupational Health, Lappeenranta, Finland.