Theoretical and experimental investigation of plumes from a circular distributed source

Apichart Chaengbamrung  
*University of Wollongong, uow_Chaengbamrung@uow.edu.au*

Paul Cooper  
*University of Wollongong, pcooper@uow.edu.au*

Peter W. Wypych  
*University of Wollongong, peter_wypych@uow.edu.au*

Ajit R. Godbole  
*University of Wollongong, agodbole@uow.edu.au*

Follow this and additional works at: [https://ro.uow.edu.au/engpapers](https://ro.uow.edu.au/engpapers)

Part of the Engineering Commons  

**Recommended Citation**  
Chaengbamrung, Apichart; Cooper, Paul; Wypych, Peter W.; and Godbole, Ajit R.: Theoretical and experimental investigation of plumes from a circular distributed source 2003, 315-320.  

Research Online is the open access institutional repository for the University of Wollongong. For further information contact the UOW Library: research-pubs@uow.edu.au
Theoretical and Experimental Investigation of Plumes from a Circular Distributed Source

Apichart Chaengbamrung, Paul Cooper*, Peter Wypych and Ajit Godbole

Faculty of Engineering
University of Wollongong
Wollongong, NSW 2522
Australia
Email*: paul_cooper@uow.edu.au

Abstract

This paper describes work in the first part of an ongoing research programme on the generation and dispersion of fume from hot metal processes, a challenging industrial ventilation problem. This phase of the study involves research into a salt solution plume generated from an area source and descending into quiescent water. This is a 'cold flow' analogue of a thermal plume ascending in quiescent air. Experiments are described whereby the velocity and concentration profiles in the plume are determined using video footage and particle-tracking software, and a conductivity probe with digital traversing mechanism, respectively. Results from a transient numerical simulation of the flow using the CFD package PHOENICS are generated and compared with experimental results. These are also used to characterise the plume with respect to density and velocity distributions.

1. Introduction

Generation and dispersion of fume from hot metal processes is an important class of industrial ventilation problem that poses several unique challenges to designers of ventilation systems [e.g. ACGIH Manual, 2001]. Generation and dispersion of fume from hot metal processes is an important class of industrial ventilation problem that poses several unique challenges to designers of ventilation systems [e.g. ACGIH Manual, 2001]. The ultimate aim of the ongoing multi-tiered research project is a fundamental investigation of the generation of fume and their subsequent dispersion via buoyancy-driven thermal plumes above molten metal surfaces [Wypych et al., 2001]. It is planned to tackle the complex overall problem in three stages: a) investigation of inverted saline plumes generated by an area source and descending into a body of quiescent clean water, b) investigation of ascending thermal air plumes from an area source, and c) investigation of generation and dispersion of fume from hot metal processes.

This paper describes the first phase of the project where a descending saline water plume is used as the 'cold flow' analogue of an ascending thermal plume. Such flows arise in many environmental and industrial settings, e.g. Colomer et al., (1999). After the well-known theoretical work of Morton, et al (1956), many researchers, e.g. Caulfield & Woods (1998), Rooney & Linden (1998) and Hunt & Kays (2001) modified that theory using different velocity and temperature or density profile assumptions (such as top hat or Gaussian). Most
of the available literature deals with the plume structure in the 'similarity' region, i.e. sufficiently far from the source. Relatively little published literature is devoted to an investigation of plume structure closer to a distributed source (e.g. Colomer, et al., 1999). Also, to date, most published theoretical and numerical analyses resort to the Boussinesq approximation, which is not valid for strong gaseous plumes with large temperature and/or density changes. This state of affairs has probably resulted in the scarcity of reliable design data for ventilation systems dealing with thermal plumes (e.g. ACGIH, 2001). Therefore, a principal objective of the present study is an experimental investigation of plume characteristics in the 'near-field' region, combined with a theoretical (numerical) analysis that does not use the Boussinesq approximation.

2. Experimental Investigation

The experiments were conducted using two geometrically similar sets of equipment, using a large cubical tank (1.4m side; source diameter 300 mm), and a smaller cubical tank (0.46m side, source diameter 105 mm). In each case, a descending plume of saline solution, was created by a flat horizontal circular source (6 mm thick fine grade sintered PTFE sheet) located just below the free surface of quiescent fresh water. The water in the smaller tank was seeded with neutrally buoyant pliolite particles of an average size of about 300 μm. Two separate optical arrangements made it possible to: a) back-illuminate the entire large tank to enable visualisation of the developing plume using the shadowgraph technique and; b) to illuminate a narrow slice of the flow field in the smaller tank passing through a plane of symmetry. The evolution of the plume was recorded with a CCD video camera and subsequently analysed using DigImage, a particle-tracking software (Dalziel, 1992). The salt concentration at different points in the flow field was measured using a specially designed conductivity probe. The tip of the concentration probe could be positioned at pre-determined positions using a computer-controlled traversing mechanism.

The salt concentration at the source varied from 5% to 20% by weight, corresponding to a range of reduced gravity at the source (from 0.42 to 1.47 m/s²). The solution flow rate at the source was adjusted to ensure that the plume was driven almost completely by density difference, i.e. jet length of the source was < 0.2mm (List, 1982).

3. Numerical Investigation

The descending plume flows in both the large and small tanks were simulated using PHOENICS CFD software (version 3.5). In both cases, a body-fitted 3D (50x50x50) mesh was used for the computational domain fitting inside the tank, and conforming to the circular shape of the source. The computational domain, like the physical domain, has impermeable vertical walls and floor. Part of the ceiling is open to the ambient (zero gauge pressure, corresponding to the free surface of water). A constant gravity source was used to account for the hydrostatic pressure condition in the tank. The standard k-ε turbulence model, with gravity correction, was used. A transient simulation was carried out, for a total duration of 10 minutes, with 10 time steps.

4. Experimental and Numerical Results

4.1 Plume width and "necking"

Figure 3 shows a comparison between an instantaneous shadowgraph image of the plume, and the corresponding plume width following image processing by DigImage. The meandering of the plume was accounted for by averaging the plume shape over 30 seconds (Figure 3b). The appearance of a 'neck' in the plume does not seem to be as distinct (εc ~ 0.28D) as indicated by Colomer et al. (1999), as after the initial contraction, the plume does not expand immediately. The neck diameter is about 0.5D, in agreement with Colomer et al. (1999).

4.2 Density Field

The density distribution data was collected by the conductivity probe over 10 seconds average time in the large experimental tank. Measurements were made in four horizontal planes approximately 10 mm, 50 mm, 100 mm, and 200 mm below the source.
It can be seen from figure 4 that density distribution in the plume from numerical simulation is in good agreement with the experimental data, at all locations except the one closest to the source. The dip in the experimental readings on the plume axis in fig 4(d) can be partly attributed to the presence of a support screw in the centre of the source disc. The discrepancy is most pronounced at the location nearest to the source. Subsequent turbulent mixing leads to better agreement between calculated and experimental data at levels further from the source.

The significance of this work lies in the fact that little detailed data has been presented by previous researchers on the temperature/buoyancy field above large distributed sources of heat/buoyancy. The results from this phase of our work will provide designers of ventilation systems and equipment located above large heat sources with data that will assist in determining local temperature fields and hence design requirements for structures and equipment. In addition, the concentration fields of contaminants released with from the buoyancy source will also be directly related to the concentration fields studied in our experiments. Hence, more accurate estimations of air cleaning requirements from situations such as hot metals processing will result.

4.3 Velocity Field

The quasi-steady velocity fields were mapped using PTV technique and DigImage software in the smaller of the two tanks. The velocity field data are presented here as moving averages taken over a 9 second duration. Figure 5 shows a comparison between the experimental and calculated fields of the vertical component of the velocity at 3 minutes from the start of the plume. It can be seen that there is reasonable agreement between experimental and computed results, except close the source, where the simulation underestimates the upward (positive) velocities. This could be due to the fact that a uniform mesh was used in the z-direction (computational cells not finer near the top boundary). This mesh was considered appropriate, as in the actual experimental arrangement, a 'solid wall' only extended a small way around the circular source, while the rest of the flow boundary was open to the atmosphere.
\[ v_r = \text{average maximum horizontal velocity} = 0.66(B_o D)^{0.5} \]  
\[ w_v = \text{vertical velocity} = 2.7(B_o D)^{0.5} \]  
\[ \text{and } B_w = \text{average plume width} = 10(B_o^2 / D)^{0.5} \] for \( z / D < 0.1 \)

In region II, where \( z > z_c \),
\[ w_v = 1.2(B_v z)^{0.5} \] for \( z / D > 0.2 \)
\[ B_w = 3.7(B_v z)^{0.5} \] for \( z / D > 0.4 \)

Applying these formulae to the present experiments, for the source condition \( B_o = 3.86 \times 10^{-5} \text{ m}^2/\text{s}^3 \), \( z_c \approx 29.4 \text{ mm} \) below the source. This suggests that the location 46.8 mm below source (fig. 5c) lies in Region II. From equation (4) at \( z = 46.8 \text{ mm} \) below the source, \( w_v \approx 0.03 \text{ m/s} \). This differs significantly from the present experimental data and numerical work, wherein \( w_v \approx 0.08 \text{ m/s} \). The same applies to the measurement location 162 mm below the source (fig 5a). Here (4) yields \( w_v \approx 0.047 \text{ m/s} \), compared to \( w_v \approx 0.12 \text{ m/s} \) in the present experiment.

5. Conclusion

This paper describes experimental and numerical studies on plumes from an area source of buoyancy. The numerical results are generally in good agreement with experimental data with regard to both velocity and density profiles. The plume from an area source shows evidence of necking with neck size \( \sim 0.5 \text{D} \), in agreement with previous research by Colomer et al., however, present measurements indicated that the position of the "neck" is not as clear as previously reported. Moreover, preliminary experimental results from the present study indicate that velocities in the plume differ significantly from the correlation suggested by the previous researchers.

6. References


