# Dilution of Dense Bottom Plumes

Petersen, O.; Larsen, Torben

*Published in:*
Proceedings of the 11th ASCE Engineering Mechanics Conference

*Publication date:*
1996

*Document Version*
Publisher's PDF, also known as Version of record

*Citation for published version (APA):*
Petersen, O., & Larsen, T. (1996). Dilution of Dense Bottom Plumes. In *Proceedings of the 11th ASCE Engineering Mechanics Conference: Fort Lauderdale, Florida, USA, May 1996*

11th International Conference of
American Society of Civil Engineers and American Society of Mechanical Engineers
Fort Lauderdale May 1996

# Dilution of dense bottom plumes

by

Ole Petersen[1,2] and Torben Larsen[1]

## 1 Introduction

The study of turbulent jets and plumes has a long tradition in the field of environmental hydrodynamics, especially in connection with design of wastewater disposal schemes in marine environments, where many studies have been reported on dilution of bouyant discharges. The present study is focused on a related but more unusual problem where the dilution in dense co-flowing bottom plumes is of concern. When e.g. industrial wastewater with large contents of salts is discharged into a turbulent current, a plume of dense negatively buoyant water may form and flow along the bed. Although the absolute density difference is small, so that dilution should be relatively rapid, the nature of such wastewater may demand that concern is given to the dilution in the vicinity of the source. The problem may also be interesting from a theoretical point of view as the steady plume represents an experimantally well-defined setup where the interactions between buoyancy and turbulence can be studied. The present study is part of a continuing work on dilution of buoyant discharges.

Studies on dense water plumes appears to be relatively scarce, but similar problems occur in other branches of fluid mechanics as e.g. spill of heavy gases (Puttock *et al.*, 1984). In figure 1 is shown a sketch of a situation where slightly denser water is discharged vertically with low speed into a turbulent stream. In the immediate vicinity of the source a complicated flow pattern is formed, where e.g. upstream intrusions and entrainment of ambient water into the plume may occur. Downstream the source a plume is formed that tend to expand laterally due to the negative buoyancy, exposing an increasing area to the ambient turbulence. Many studies have demonstrated that vertical mixing across a density interface is dampend (Turner, 1973), thus the mixing will be slow and the plume character of the flow can be quite persistent downstream.

Below is described a series of experiments and modelling on an idealised situation where a steady negatively buoyant plume is discharged vertically through a single port into a fully developed turbulent current.

## 2 Experiments

The objectives of the experimental set up has been to obtain a well defined ambient flow with known turbulence and mixing and a steady discharge of denser water. The

---

[1] Aalborg University, Department of Civil Engineering, Sohngaardsholmsvej 57, DK9000 Aalborg
[2] International Centre for Computational Hydrodynamics (ICCH), Danish Hydraulic Institute, Agern Alle 5, DK2970 Hørsholm

experimental facility consists of a 20 m long and 1.5 m wide flume with a smooth painted bottom that recirculate approximately 10 $m^3$ of water with a temperature at 18 °C. . A dense plume is obtained by discharging cold (8 °C.) water from a reservoir at a constant, metered rate through a circular 28 mm diameter pipe located 12 m downstream in the flume and mounted flush with the bottom. The relatively long inlet section and a honeycomb assures a well defined boundary layer turbulence in the measuring section.

The instrumentation consist of a vertical rake of 8 thermocouples with 5 to 30 mm spacing, that can traverse the plume in different downsteam cross section. The thermocouples has the cold junction placed in an insulated reservoir and are connected to a computer that controls the traversing and scanning of the thermocouples. With the present amplification the resolution of the measurement is 0.02 °C. This method is well suited here because only temperature differences are important.

The plume is in each cross section characterised by three parameters, the excess mass $m$, a height $h$ and a width $w$, defined from numerical integration of the measured temperature distributions as

$$m = \int\int \Delta\rho dy dz \qquad h^2 = \frac{1}{m}\int\int \Delta\rho z^2 dy dz \qquad w^2 = \frac{1}{m}\int\int \Delta\rho y^2 dy dz$$

where $y$ and $z$ are cartesian coordinates and $\Delta\rho$ is found from an equation of state.

A number of 8 experiments are made with combinations of discharges from 3.3 $10^{-5}$ to 1.6 $10^{-4}$ $m^3/s$, initial temperature differences from 6.0 to 7.6 °C and ambient velocity from 7.0 to 20.0 cm/s. The water depth is kept constant at 17.0 cm using a downstream weir.

## 3 Models of dense plumes

### 3.1 Integral model

To guide the analysis of the experiments a simple integral theory is developed that can describe the plume in terms of the downstream development of height and width. Briefly, the theory is based on the assumptions that the steady plume follows the ambient mean flow, thus the flux of buoyancy through a crossection is constant $U_a h w \Delta\rho$. The celerity of the lateral expansion of the plume is assumed to be $V_f = \alpha\sqrt{g'h}$ where $g' = g\Delta\rho/\rho$ is reduced gravity and $\alpha$ a calibration constant. Combining this with continuity and parameterizing the turbulent mixing as a diffusion process debelopment equations for $h$ and $w$ can be given as

$$\frac{dw}{dx} = \frac{V_f}{U_a} + \frac{K_y}{U_a w} \qquad \frac{dh}{dx} = -\frac{h}{w}\frac{V_f}{U_a} + \frac{K_z}{U_a h}$$

where $K_y$ and $K_z$ are turbulent diffusion coefficients. The attenuation of vertical diffusion by buoyancy is assumed to depend on the stability through a Richardson number $R_{io}$ as $K_z = K_{zo}/(1+\beta R_{io})$ where $R_{io} = g' h / u_f^2$, $u_f$ is friction velocity and $\beta$ an empirical constant. $K_{zo}$ and $u_f$ are related to the undisturbed ambient flow.

## 3.2 A numerical model

A 3 dimensional numerical model has been developed, where the experimental results can be incorporated into a more general frame. The model is based on the 3 steady hydrodynamical eauations coupled to a transport equation for buoyancy. A key point is the description of turbulent mixing, where a viscosity model based on a k-ε closure which includes buoyancy effects is used. The equations are discretised on a staggered cartesian grid and the solution is based on CFD standard methods as the Simple method.

## 4 Results

In figure 2 is displayed the development of the relative plume width for all experiments and also from the numerical model. The scaling corresponds to an immiscible plume (Weill and Fischer, 1974) and the full line should thus fall below the measured points.

The primary result of the experiments is a calibration of the two constants in the integral model. As part of this calibration the undisturbed ambient flow has been characterised by velocity measurements and varuois tracer experiments from which the bed friction and the vertical diffusivity can be estimated. From the set of measured plume widths and heights and using a numerical optimisation procedure the values of the two constant that minimise the error in dilution is estimated to $\alpha=1.2$ and $\beta=1.0$.

In figure 3 is shown one example of calculated isoterms calculated with the numerical model. Comparison with the measurements indicate that the distribution is quite realistic. Because there is a relatively large random element in this type of experiments a more quantitative comparison with the numerical model is made through the integral model. Using a series of numerical experiments, corresponding to the phisycal experiments, a parallel calibration of the integral model is made. This calibration gives two different values for the constants $\alpha=1.2$ and $\beta=1.2$, indicating that the front celerity is well reproduced while the mixing in the numerical model is somewhat to large.

## Acknowledgements

Part of this work is founded by the Danish National Research Foundation.Their Support is greatly appreciated.

## References

Petersen, O.(1994). Laboratory and numeriical experiments on the dilution of buoyant surface plumes. In : Davies, P. A. and Neves, M. J. (eds.) *Recent dvances in the Fluid Mechanics of Turbulent Jets and Plumes.* Kluwer Academic Publishers, Dordrecht, the Netherlands, 223-236.

Rodi, W. (1987). Examples of calculation methods for flow and mixing in stratified fluids. *J. Geophys. Res.*, **92**(C5), 5305-5328.

Turner, J. S. (1973). *Buoyancy effects in fluids.* Cambridge University Press, UK.

Weill, F. and Fischer, H. B. (1974). Effect of stream on heater water plumes. *J. Hydraul. Div.;* ASCE, **100**(HY7), 951-970

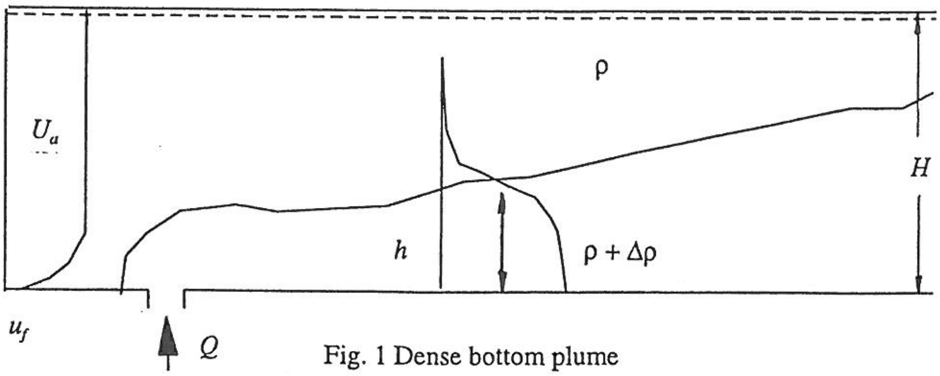

Fig. 1 Dense bottom plume

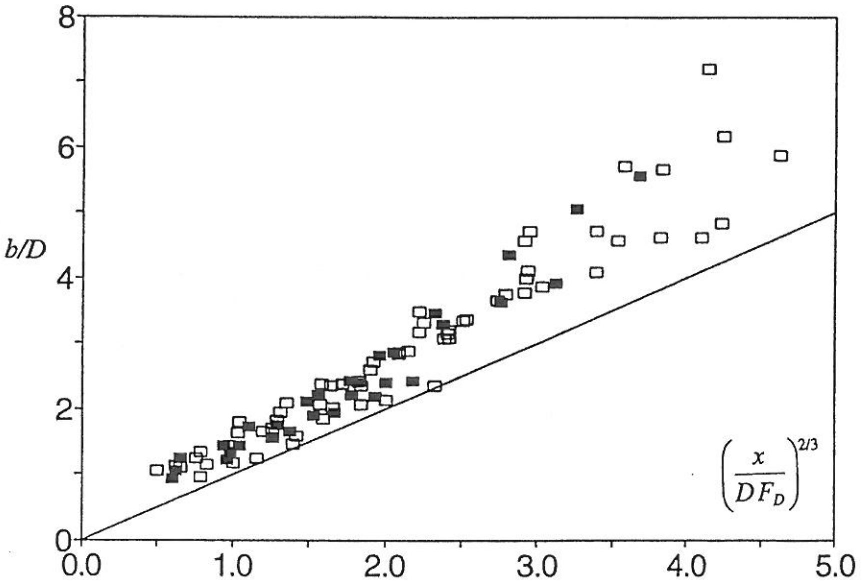

Fig. 2 Downstream development of relative plume width. Open squares are from the experiments, filled squares from the numerical model. The full line follows data from (Weill and Fischer, 1974)

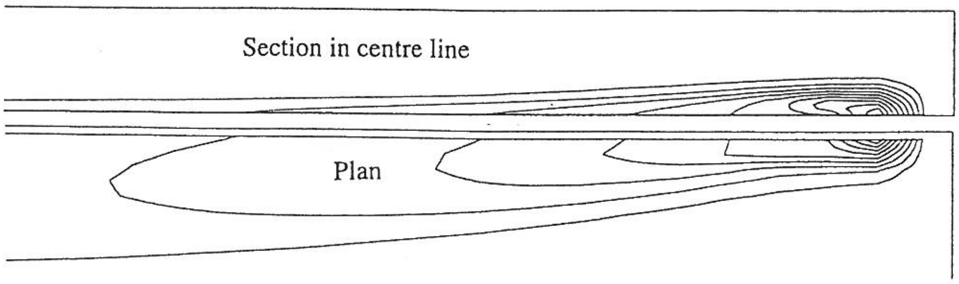

Fig. 3 Isoterms calculated with the numerical model. Run # 11 : Ua = 0.1 m/s, $\Delta T_o$ = 12.3 °C, $H$= 0.17 m, $Q_o = 8.0\ 10^{-5}\ m^3/s$.