Numerical Modelling of Jets and Plumes

a civil engineering perspective

Larsen, Torben

Published in:
Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes : Proceedings of the NATO Advanced Research Workshop on Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes

Publication date:
1994

Document Version
Early version, also known as pre-print

Citation for published version (APA):
Larsen, T. (1994). Numerical Modelling of Jets and Plumes: a civil engineering perspective. In M. J. V. Neves, & P. A. Davies (Eds.), *Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes : Proceedings of the NATO Advanced Research Workshop on Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes: Viana do Castelo, Portugal, June 28 - July 2, 1993* Kluwer Academic Publishers.

NUMERICAL MODELLING OF JETS AND PLUMES - A CIVIL ENGINEERING PERSPECTIVE

TORBEN LARSEN
Department of Civil Engineering
Aalborg University
Sohngaardsholmsvej 57
DK-9000 Aalborg
Denmark

ABSTRACT. An overview on numerical models for prediction of the flow and mixing processes in turbulent jets and plumes is given. The overview is structured to follow an increasing complexity in the physical and numerical principles. The various types of models are briefly mentioned, from the one-dimensional integral method to the general 3-dimensional solution of the Navier-Stokes equations. Also the predictive capabilities of the models are discussed. The presentation takes the perspective of civil engineering and covers issues like sewage outfalls and cooling water discharges to the sea.

1 Introduction

1.1 BACKGROUND

This overview on numerical modelling of jets and plumes is, according to the objectives of behind the workshop, thought to be reflective more than exact and mathematical. It attempts to discuss and show the connection between the physical and the numerical aspects of the subject. It also tries to compare the principles mentioned with each other. For these reasons, the overview is broad and far from any complete and general reference to the subject.

To identify the various models, some of the basic equations for the models have been listed. But this listing is not complete in any sense and important points like, for example, equations for the boundary conditions are not included.

1.2 DEFINITION OF NUMERICAL MODELS

A quantitative prediction of the behaviour of a physical system necessitates some numerical operations. Nowadays such operations will almost always be performed on a computer. Nevertheless, not all quantitative predictions of jet and plumes should be classified as numerical models. So, attempts to distinguish clearly between analytical and numerical models seems difficult.

Properly, the most clear distinction between numerical and analytical descriptions is the degree of generality of the solutions. The result of an analytical analysis is often a functional relation given in a formula or graph, which covers a range of dependent and independent variables such as, for example, velocities, densities etc. often in combinations in non-dimensional numbers. In contrast the numerical model gives the specific solution in terms of numbers based on defined initial- and boundary conditions.

Likewise, and in contrast to the analytical models a distinguishing feature is the generality of the numerical model itself and not the result it delivers. The numerical model can be executed over and over again with various boundary and initial conditions.

So, from the author's point of view a numerical model can be defined and characterized by a sequence of repeated numerical operations, which by advantage can take place in a computer and which in principle only give one specific result of the general solution.

1.3 EXPERIMENTAL VERSUS NUMERICAL WORK

The fast development of computers and numerical methods during the last three decades has developed a gap in hydraulics research between experimental and numerical work. Experimentalists often want to present their results by clear analytical methods where generalization is obtained by the use of similarity principles expressed in relations between dimensionless numbers. From the opposite front much development of numerical models takes place without any experimental verification and without any feeling for the essential question of how an experimental validation of the modelling can take place in respect to measuring technique, resources and accuracy.

From the author's point of view, it can be acceptable in special cases to set up and apply a numerical model combining existing experience to a non-verified conglomerate of 'the best actual knowledge' for a specific problem. But there seems to exist a dangerous tendency to perceive the application itself of such a model as a kind of recognition or verification of the model which justifies a further increase in model complexity. A kind of higher order of lack of verification then occurs. This type of numerical modelling can be found in the grey area between research and commercial use of numerical models.

Despite the strong commercial influence on the area, serious research fortunately takes place in a number of organizations, and a dramatic increase in the general use of numerical models is to be expected in the coming decade.

1.4 THE ENGINEERING PERSPECTIVE

From a scientific viewpoint some numerical models of jets and plumes are definitely more preferable to others, as discussed later in more detail. But in respect to civil engineering design and decision making, jet and plumes are only a specific problem in a broader context. Therefore, jets and plumes are almost always modelled together with ambient currents and turbulence, often with complex boundary conditions. The total accuracy of the modelling then depends not only on the best possible description of the jet and plume. The engineering perspective is to choose not only the best but also the most reliable and accepted principle for the actual situation.

Many examples of this conflict between the scientific and the engineering perspective can be given. One is the design of long sea outfalls in relation to modelling of the water quality in the surroundings. In this case jets and plumes will be relevant for the local dilution - the initial dilution in the rising part and the dilution in the first stage of the following horizontal transport. But for the far field, where the transport and dilution are controlled by the ambient current and dispersion, it is a general experience that the initial stage has only marginal influence on the final and overall dilution. With this background and in the engineering perspective it is not only acceptable, but also more correct to use a simple and less accurate description of the initial dilution in the model, and allocate the resources to more significant aspects of the problem.

2 Integral models of submerged jet and plumes

2.1 GENERAL

Integral jet and plume models are widely used in engineering practice. The integral principle is well known from a variety of practical application of the integrated momentum and energy equations in pipeline flows, in free surface flows etc. This integral principle has been used for several decades to produce analytical as well as numerical solutions to jet and plume problems.

Numerical modelling with integral models began in the late sixties and in the work of Fan (1967) and Fan and Brooks (1969) the ideas which have lead to the situation to today were already established. Very significant improvement on a better formulation of the obtaining entering physical conditions has since been achieved, but the numerical principle has not been similarly developed. Of important recent work, the papers of Lee and Cheung (1990) and Wood (1993) should be mentioned as examples; more complete references can be found therein.

As a result of this research during more than two decades a number of reliable models are now available.

2.2 BASIC EQUATIONS

Integral models can describe either 2-dimensional slot jets or axi-symmetric jets with 2- or 3-dimensional trajectories.

Only the axi-symmetric jets in 2 dimensions will be discussed here because the main physical conditions of the other types are covered this way. The main assumption is that all velocity distributions $u(s,r)$ and all concentration distributions $c(s,r)$ are self-similar and follow the Gaussian distribution:

$$u(s,r) = u_m(s)\exp(r^2/b^2) \tag{2.1}$$

$$c(s,r) = c_m(s)\exp(r^2/\lambda^2 b^2) \tag{2.2}$$

where s is a co-ordinate along the jet trajectory, r is distance from the centerline of the jet, b the jet half width, λ is a constant, u_m and c_m are centerline velocity and concentration. A state equation, which gives the relation between concentration c and density ρ, is assumed to be known. On this background the integral parameters *volume flux* Q, *momentum flux* M and *buoyancy flux* B can be defined as

$$Q(s) = \int_A [u_a(s)\cos\theta + u(s,r)]dA \tag{2.3}$$

$$M(s) = \int_A \rho(s,r)[u_a(s)\cos\theta + u(r,s)]^2 dA \tag{2.4}$$

$$B(s) = \int_A \Delta\rho(s,r)[U_a(s)\cos\theta + u(r,s)]dA \tag{2.5}$$

where u_a is the ambient velocity, $\Delta\rho$ the density difference and A is a plane perpendicular to the centerline. The momentum flux $M(s)$ is a vector in the centerline direction. The integral equations for a jet can now be formulated as the local conservation of volume, mass, horizontal momentum and vertical momentum. Details are not given here but after integration over the cross-section of the jet the integrals appear as:

Conservation of volume

$$\frac{d}{ds}[\pi b^2(2u_a\cos\theta + u_m)] = q_e \tag{2.6}$$

Conservation of mass

$$\frac{d}{ds}[b^2(u_m+(1+\lambda^2)u_a c_m \cos\theta] = -\frac{1+\lambda^2}{\lambda^2}\frac{dc_a}{ds}[b^2(u_m+u_a\cos\theta)] \tag{2.7}$$

Conservation of density deficit

$$\frac{d}{ds}[b^2(u_m+(1+\lambda^2)u_a \cos\theta\Delta\rho_m] = -\frac{1+\lambda^2}{\lambda^2}\frac{d\rho_a}{ds}[b^2(u_m+u_a\cos\theta)] \tag{2.8}$$

Conservation of horizontal momentum

$$\frac{d}{ds}[\pi\rho_a b^2(2u_a\cos\theta+u_m)^2\cos\theta] = 2F_D\sin\theta+2\rho_a u_a q_e \tag{2.9}$$

Conservation of vertical momentum

$$\frac{d}{ds}[\pi\rho_a b^2(2u_a\cos\theta+u_m)^2\sin\theta] = 2F_B-2F_D\cos\theta \tag{2.10}$$

Geometric equations

$$\frac{dx}{ds}=\cos\theta \quad \text{and} \quad \frac{dz}{ds}=\sin\theta \tag{2.11}$$

The interaction with the ambient flow is described by the entrainment q_e in the continuity equation, the drag forces F_D and the buoyancy force F_B.

The drag force per unit length between the jet and the ambient flow is assumed to follow the well-known equation for drag on solid bodies:

$$F_D = c_D\frac{1}{2}\rho_a(u_m\cos\theta-u_a)^2\pi b^2 \tag{2.12}$$

The drag coefficient c_d is found empirically. The order of magnitude is 1.0.
The buoyancy force F_B per unit length can be determined from

$$F_B=\pi b^2\Delta\rho_m g \tag{2.13}$$

2.3 ENTRAINMENT COEFFICIENTS

To illustrate how a numerical model based on the integral principle will precede only some simple examples of equations for entrainment coefficients will be shown in this section. For stagnant ambients the entrainment coefficient according to Morton *et al* (1956) is defined by

$$q_e=\alpha\, 2\pi b\, u_m \tag{2.14}$$

A suggestion for the entrainment coefficient α is given by Hirst (1971):

$$\alpha = 0.057+\frac{0.97}{F_l}\sin\theta \tag{2.15}$$

in which F_l is the local densimetric Froude number, defined as

$$F_l=\frac{u_m}{\sqrt{\frac{\Delta\rho_m}{\rho_a}g\,b}} \tag{2.16}$$

In flowing turbulent ambients several hypotheses have been presented. Recently, a generalized entrainment function has been proposed by Lee and Cheung (1990) who separated the total entrainment into a shear entrainment q_{es} and a forced entrainment q_{ef}. The forced entrainment appears due to the convection of the jet and should not be understood as entrainment in a physical sense. The equations proposed are

$$q_{es} = 2\pi\alpha(u_m - u_a \cos\theta) \tag{2.17}$$

$$q_{ef} = u_a\left(2b\sqrt{1-\cos^2\theta} + \pi b \Delta b \cos\theta + \frac{1}{2}\pi b^2 \Delta\cos\theta\right) \tag{2.18}$$

2.4 TRANSITION FROM THE RISING PLUME TO THE HORIZONTALLY FLOWING PLUME

A free rising buoyant plume will either be trapped in a certain height, where the plume reaches the same density as the ambient, or will continue to the surface. Several flow patterns can exist for this situation as described by Wood (1993). But, in both cases a stationary upstream wedge may be formed when the ambient flow velocity is low. This is normally the most critical pattern in respect of environmental considerations.

Unfortunately, the dilution in this transition from a rising plume to a horizontally flowing plume is not very well described in the literature and, accordingly, the dilution in this stage is not included in the numerical models.

2.5 NUMERICAL SOLUTIONS OF INTEGRAL MODELS

For 2-dimensional trajectories we have 7 first order differential equations but as dx and dz can be determined directly there remains only 5 to be solved, which are u_m, b, c_m, $\Delta\rho_m$ and θ.

The solution technique for this set of coupled equations does not contain any important problem. A standard method like, for example, the well-known fourth order Runge-Kutta, can be employed. The only important point is to choose increments of Δs so small that the solution is independent of Δs.

Boundary conditions are always important to consider. Near the outlet a *zone of establishment* can be incorporated in the model to give the first set of jet characteristics from where the self-similar principle are valid.

3 Integral Models of Surface Jets and Plumes

3.1 GENERAL

Due to the high discharges, surface discharge of cooling water from power production often takes place through horizontal outlets. The temperature deficit of the discharge corresponds to a density deficit, which causes an increase in the lateral spreading of the jet.

To cover the complete description from the outlet through the jet region to the plume region the calculation is split into the following stages:

- *Zone of flow establishment.* In this zone the flow profile changes from rectangular to the Gaussian profile.
- *Jet with bottom contact.* If the zone of flow establishment is associated with bottom contact the jet will normally proceed some distance before the bottom contact is lost.

- *Free jet.* The free jet is here flowing with an excess velocity compared to the ambient flow.
- *Intermediate stage.* In this stage the plume finally loses the excess velocity and density.
- *Passive plume.* Here the transport and dispersion are controlled by the ambient conditions.

3.2 BASIC ASSUMPTIONS

The integral approach is based on integrating the conservation of volume, mass and momentum across the jet or plume cross section. Furthermore, the excess velocities, temperatures and concentrations are assumed Gaussian:

$$U = U_a \cos\theta + U_m \exp\left(-\frac{\eta^2}{B_j^2}\right)\exp\left(-\frac{\xi^2}{H_j^2}\right) \tag{3.1}$$

$$C = C_m \exp\left(-\frac{\eta^2}{B_c^2}\right)\exp\left(-\frac{\xi^2}{H_c^2}\right) \tag{3.2}$$

Where U is the velocity inside the jet, U_a is ambient velocity, θ is the jet centerline angle with the coastline, U_m is the centerline velocity, C_m is the centerline concentration, B_j, H_j are the characteristic width and depth respectively of the velocity profile, B_c, H_c are corresponding dimensions of the concentration profile, C is the excess concentration and s, η, ξ are the local co-ordinates along the centerline. The general and full equations are quite complex and space limitations do not allow them to be written here. In fact the structure of the equations follows the equations for the submerged jet already shown. Details can be seen in Møller *et al.* (1990) and in Hansen (1991) which give the background for the model named *NEWJET* developed by the *Danish Hydraulic Institute*. This model includes 11 first-order differential equations:

- *Continuity.* Continuity of volume, local continuity of matter, global continuity of matter, global continuity of temperature and global continuity of salt.
- *Momentum.* Momentum along centerline, momentum perpendicular to centerline, buoyant frontal spreading.
- *Geometric equations.*
- *State of water.* Relation between density, temperature and salinity.

In the equations are included several other effects like heat exchange with the atmosphere, the effect of wind on the vertical mixing etc. An impressive amount of empirical experience is incorporated in this type of model, including several examples of field validations.

But from an a numerical viewpoint the model is simple and could in principle be run in a spreadsheet program. The equations are solved by a standard predictor-corrector method.

The ambient current field will normally be determined by a 2-dimensional hydrodynamic model, covering an area much larger than covered by the plume model. The plume model will often deliver the results to a far-field hydrodynamic and transport/dispersion model, which again decribes the excess temperature in a larger area. Superposition of plumes in the integral model is of course not possible, but the far-field model can receive results from several independent plumes.

3.3 CO-FLOWING BUOYANT SURFACE PLUMES

In shallow waters surface plumes of diluted sewage from long sea outfall diffusers will form during low ambient flow. The flow is characterized by the lack of excess velocity between the plume and the ambient flow in the longitudinal direction. This problem has been discused by Weil and Fisher (1974), Petersen and Larsen (1989), Petersen and Larsen (1990) and Petersen (*q.v.*).

Following Petersen (*q.v.*) the plume flow can be described with a simplified integral model for the development of the plume height h and half width b according to the following coupled equations:

$$\frac{dh}{dx} = -\frac{h}{b}\frac{V_f}{U} + \frac{K_z}{Uh} \tag{3.3}$$

$$\frac{db}{dx} = \frac{V_f}{U} + \frac{K_y}{Ub} \tag{3.4}$$

$$hb\Delta\rho = h_0 b_0 \Delta\rho_0 \tag{3.5}$$

where x is the longitudinal co-ordinate, V_f is the front velocity, U is the ambient velocity, K_z,K_y are the vertical and transverse dispersion coefficients respectively and $h_0 b_0 \Delta\rho_0$ refers to the initial condition. The vertical mixing was assumed to be damped because of the density gradient. The front velocity and the vertical dispersion coefficient were found from laboratory experiments and turbulence modelling to give

$$V_f = \alpha\sqrt{gh\frac{\Delta\rho}{\rho}} \tag{3.6}$$

$$\frac{K_z}{K_{z0}} = \frac{1}{(1+\beta\mathbf{R}_{i0})} \qquad where \qquad \mathbf{R}_{i0} = \frac{gh\Delta\rho}{\rho u_f^2} \tag{3.7}$$

where K_{z0} is vertical dispersion coefficient under neutral conditions in the depth h below the surface, $\mathbf{R}_{i0}$ a bulk Richardson number, u_f is friction velocity in the ambient flow; the empirical constant α was determined to be 1.1 and β to be 3.3.

The formulation will give asymptotically the correct solution for the neutral plume when the density difference gradually vanishes.

4 Neutral Plumes

Plumes which passively follow the ambient flow without generating any change in the flow field should be mentioned here because their use has significant interest for many applications.

1-, 2- and recently also 3-dimensional models based on the transport/dispersion equation have been applied for a variety of applications since the beginning of the seventies. The models often cover the solution of several simultaneous and coupled equations for various chemical and biological constituents. These so-called water-quality models are often coupled to flow-field models of various complexity.

The governing transport/dispersion equation can be written in tensor notation as

$$\frac{\partial C}{\partial t} + U_j\frac{\partial C}{\partial x_i} = \frac{\partial}{\partial x_j}(D_i\frac{\partial C}{\partial x_j}) + S \tag{4.1}$$

where C is the time averaged concentration, D_i is the set of dispersion coefficients and S is a source term.

The advantage of this description is that it easily can fit into the same spatial grid as used by the flow model. The disadvantage is that numerical dispersion arises from the discretization of the second term, the advective change. This problem is enhanced by the fact that the concentration gradients normally are much larger than the gradients of the flow field. Thus, when the hydrodynamic model is coupled to the transport model on the same grid, the grid size is normally restricted by the transport model.

In order to overcome the problem of numerical dispersion, the application of an alternative method based on the random walk principle has grown widely. The principle is to keep track of the positions of a large number of particles. The movement of each particle for each time step consist of an advective translation plus a random jump to represent the dispersion. Furthermore, other aspects like a fall velocity can be included. An example of the principle for neutral sea outfall plumes was given by Larsen (1983). Today the major international consultants apply the method in practice. An example is seen later.

The x co-ordinate of the position of a particle at the new time step can be written as

$$x_{i+1} = x_i + U_x \Delta t + \sqrt{2D_x \Delta t} \cdot rnd(0,1) \tag{4.2}$$

where x_{i+1} is new co-ordinate, x_i is old co-ordinate, U_x is ambient velocity, D_x is dispersion coefficient, Δt is the time step and *rnd(0,1)* is a Gaussian distributed random number with average *0* and standard deviation *1*.

The ambient flow velocity will normally be calculated by a numerical model based on the control-volume principle as mentioned in the next section.

5 Turbulence models

5.1 GENERAL

A short review of turbulence modelling is given in this section. A more complete overview was given by Rodi (1984), who also (Rodi, 1987) presents important points in respect of modelling of flow and mixing in stratified flows. A broad discussion on the applicability of turbulence models can be found in ASCE (1988).

Seen from the outside, turbulence modelling looks difficult and unapproachable. This is certainly also the case from the mathematical and numerical point of view. But most complexities are generated from the classic contrast between our preference for using an Eulerian co-ordinate system instead of the more physical related Lagrangian (co-flowing) reference. The physical assumptions behind the turbulence models are much simpler than the complicated mathematical environment necessary for the description of the flow field in fixed co-ordinates.

An interesting development concerning the mathematical and numerical modelling of 2- and 3-dimensional flows is the appearance of software packages which do most of the hard numerical work by solving the Navier-Stokes and the transport equations. These so-called equation-solving programs give the user freedom to add source terms in the conservation equations and in this way build specific models for a variety of applications like turbulence models, ground water models, combustion models etc. Furthermore, these models take care of input and output of data often including advanced graphical methods.

5.2 NAVIER-STOKES EQUATIONS

Basically the Navier-Stokes equations are derived from Newton's second law and the law of mass conservation. Although the development of the equations seem exact it is worthwhile mentioning that a number of simplifying and not always obvious assumptions have been made to achieve their formulation. An example of such an assumption is the definition of pressure as the average of the 3 normal stresses, which indicates that troubles may show up when non-Newtonian and perhaps anisotropic fluids are modelled with the equations. Nevertheless, the equations have proved their validity in a wide spectrum of applications.

In the basic formulation it is seen that the equations allow the transfer of shear stresses only as viscous forces, which of course is the only way shear stresses can be transferred in Newtonian fluids, by definition. In turbulent flow the smallest eddies are characterized by the Kolmogorov length scale, the turbulent micro scale, given by $\lambda = (\nu^3/\varepsilon)^{1/4}$, where ν is the kinematic viscosity and ε is the dissipation. The size of the Kolmogorov length scale in practical flow problems varies but an order of magnitude will be a part of a millimetre. To achieve a full resolution of velocities and forces a grid size less than the Kolmogorov length scale and/or the thickness of any viscous sublayers at the boundaries is needed. As discussed later, the number of grid points and time steps in such a description lies several orders of magnitude above the capabilities of today's and tomorrow's computers capabilities, so this viewpoint is not useful in numerical modelling. To overcome this problem the Navier-Stokes equations have to be integrated up to a scale which corresponds to the computer's capacity in respect of memory and speed.

The dominating issue in turbulence modelling is the question of averaging in time and space. The maximum number of grid points in 3-dimensional modelling lies in the order of 100^3. Although this seems to be a high number, even a small physical model can easily give a much higher resolution.

To integrate the basic Navier-Stokes equations with respect to time Osborne Reynolds proposed to split the instantaneous velocities, pressures etc. into an average part U and an fluctuating part u' .

$$u = U + u' \qquad where \qquad U = \frac{1}{T}\int_0^T u\,dt \tag{5.1}$$

Now, assuming

- incompressible fluid,
- density differences only appear in the gravity term (the Boussinesq approximation),
- each term is averaged over a time T longer than the turbulent macro scale,

the equations in tensor notation read:

Continuity equation

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{5.2}$$

Momentum equations:

$$\frac{\partial U_i}{\partial t} + U_j\frac{\partial U_i}{\partial x_i} = -\frac{1}{\rho_r}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\overline{u'_i u'_j} + \nu\frac{\partial U_i}{\partial x_j}\right) + g_i\frac{\rho_r - \rho}{\rho_r} \tag{5.3}$$

Soluble transport equation (concentration, temperature, turbulence):

$$\frac{\partial C}{\partial t} + U_j\frac{\partial C}{\partial x_i} = \frac{\partial}{\partial x_j}\left(-\overline{c'u'_j} + \lambda\frac{\partial C}{\partial x_j}\right) \tag{5.4}$$

The 9 terms $\overline{u'_i u'_j}$ represent the momentum fluxes by the turbulent fluctuations and are known as the Reynolds stresses. The 3 transport terms $\overline{u'_i c'}$ are the turbulent heat or mass fluxes.

For high Reynolds numbers the turbulent stresses and fluxes are orders of magnitude larger than viscous stresses and fluxes. The key-problem in turbulence modelling is how these terms are found. It can be shown that these correlations cannot be derived from the Navier-Stoke equation without the appearance of new higher order correlations. In the following some of the most well-known closures of this fundamental problem are outlined.

5.3 EDDY VISCOSITY CONCEPT

This first approach to describe turbulent shear stresses was proposed by Boussinesq, who appealed to simularity to laminar flow and defined an eddy viscosity ν_t by

$$\tau_t = -\rho\overline{u'w'} \equiv \rho\nu_t\frac{\partial U}{\partial z} \tag{5.5}$$

where τ_t is the turbulent shear stress. An equation for the normal stress also exists but is not shown here.

For the mass flux, the following equation is used

$$F_t = -\overline{w'c'} \equiv D_z\frac{\partial C}{\partial z} \tag{5.6}$$

5.4 CONSTANT EDDY VISCOSITY MODELS

The assumption of a constant eddy viscosity has had some practical applications in the modelling of the flow in receiving waters, for example in early lake and reservoir modelling where the eddy viscosity was assumed to be proportional to the wind friction velocity. But for jets and plumes this assumption is not of much relevance.

5.5 ZERO EQUATION MODELS - LENGTH SCALE MODELS

For dimensional reasons, the eddy viscosity ν_t has to be proportional to a velocity scale V and a length scale L:

$$\nu_t \sim VL \tag{5.7}$$

Prandtl made the assumption (Schliching, 1969) that the velocity scale should be proportional to the gradient of the mean velocity as

$$V \sim L\frac{\partial U}{\partial z} \tag{5.8}$$

Combining these equations leads to the so-called length scale formulation for the eddy viscosity:

$$\nu_t = L^2\left|\frac{\partial U}{\partial z}\right| \tag{5.9}$$

5.6 ONE-EQUATION MODELS - CONSERVATION OF TURBULENT KINETIC ENERGY

The next step in model complexity was to relate the eddy viscosity to some scalars containing information on the turbulence characteristics and then describe the transport of these scalars by a kind of conservation equation. Prandtl and Kolmogorov [Rodi, 1984] suggested to use the turbulent kinetic energy k, defined by

$$k \equiv \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \tag{5.10}$$

as the square of the velocity scale. Based on this assumption the eddy viscosity yields

$$\nu_t = c'_\mu L \sqrt{k} \tag{5.11}$$

Here L is the length scale of the most energy containing eddies and the c'_μ is an empirical constant.

The k-equation can be derived from the Navier-Stokes equations, but here the closure problem shows up because unknown correlation terms appear in the diffusion and dissipation terms. For the diffusion term a gradient flux assumption is made and for the dissipation an assumption based on dimensional analysis is employed (Rodi, 1984). This leads to the k-equation:

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + \nu_t \frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + \frac{g_i}{\rho_r}\frac{\nu_t}{\sigma_t}\frac{\partial \rho}{\partial x_i} - \frac{c_D k^{3/2}}{L} \tag{5.12}$$

Models based on this equation and the Navier-Stokes equations are often denoted as one-equation models. In such models the length scale has to be understood as a calibration parameter. Many useful results in boundary flows and in sediment transport have been achieved by this type of model.

As seen, the length scale appears in the eddy viscosity and in the dissipation terms. But the length scale itself (for example, the width of the jet or plume) will often be the goal of the modelling. So in respect to jets and plumes, it is obvious that the principle is not suitable.

5.7 TWO-EQUATION MODELS - k-ε MODELS

In order to overcome the need for a more general description, a model for the length scale is necessary. From dimensional arguments it was suggested to let the length scale be derived as

$$L \cong \frac{k^{3/2}}{\varepsilon} \tag{5.14}$$

where ε is the local dissipation of the turbulent kinetic energy. Immediately it seems self-contradictory to define a length scale on a local condition, but k and ε are not independent variables and they cannot take values from independent reasons. Experience showed that this assumption nevertheless led to a satisfactory result for a number of applications. Now the eddy-viscosity can be found from

$$\nu_t = c_\nu \frac{k^2}{\varepsilon} \tag{5.15}$$

An equation for ε can be found from the Navier-Stokes equations (Daly and Harlow, 1970). Some simplifying assumptions are necessary. The coupled k-equation and dissipation equations now read

$$\frac{\partial k}{\partial t}+U_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right)+\nu_t\frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)+\frac{g_i}{\rho_r}\frac{\nu_t}{\sigma_t}\frac{\partial \rho}{\partial x_i}-\varepsilon \tag{5.16}$$

$$\frac{\partial \varepsilon}{\partial t}+U_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right)+c_{1\varepsilon}\frac{\varepsilon}{k}\nu_t\frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)+c_{3\varepsilon}\frac{g_j}{\rho_r}\frac{\nu_t}{\sigma_t}\frac{\partial \rho}{\partial x_j}-c_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{5.17}$$

The constants in the k-ε-model are found from extensive examinations of free turbulent flows and wall flows. Results are most sensitive to the values of $c_{1\varepsilon}$ and $c_{2\varepsilon}$. For example, a 5 % change of one of these constants will change the spreading rate of a jet by 20 %. The generally most accepted values of the constants were given by Launder and Spalding (1974), as seen in the following table.

c_ν	$c_{1\varepsilon}$	$c_{2\varepsilon}$	c_k	c_ε
0.09	1.44	1.92	1.0	1.3

The k-ε models have achieved wide applications not only in research but also in engineering practice. For jets and plumes, many successful examples of applications can be found (see for example, Chen and Rodi, 1975; Patankar *et al.*, 1977; and Larsen *et al.*,1990).

As stated by Rodi (1984) complete universality of the mentioned constants should not be expected. For stagnant surroundings the rate of spread of an axi-symmetric jet was overpredicted by as much as about 30 %. To compensate for this Rodi (1984) recommended a modification of the c_ν constant to be a function of the deceleration along the axis of the jet.

The advantage of the use of 2- and 3-dimensional turbulence models is their ability to predict complex flows with more than one characteristic flow type; for example, a jet near a wall with a certain ambient flow. For such complex flows specific modifications as mentioned above are not possible if all aspects should be satisfied. Fortunately, the experience shows more reliable results for complex flows.

An obviously weak point of the k-ε models is the isotropic description of the turbulence, or more correctly of the eddy viscosity. In order to improve the predictive capability of the models and to achieve a higher degree of universality higher order closures for the turbulence has now entered not only hydraulic research but also the engineering world.

5.8 REYNOLDS STRESS MODELS

The isotropic eddy viscosity concept mentioned in the previous section has its limitation, especially in flows with directional influence on turbulence due to gravitational forces. This section briefly discuss the Reynolds stress models, which employ transport equations for the Reynolds stresses $\overline{u'_iu'_j}$ and the turbulent fluxes $\overline{u'_ic'}$. Models of this type are often denoted second-order-closure schemes and the literature gives many examples of proposed formulations. The following is taken from Gibson and Launder (1978) also referred by Rodi (1987). The 6 Reynolds stress equations read

$$\frac{\partial \overline{u'_i u'_j}}{\partial t} + U_l \frac{\partial \overline{u'_i u'_j}}{\partial x_l} = \qquad (\textit{local change} + \textit{convective change} =)$$

$$c_s \frac{\partial}{\partial x_l}\left(\frac{k}{\varepsilon}\overline{u'_k u'_l}\frac{\partial \overline{u'_i u'_j}}{\partial x_k}\right) \qquad (\textit{diffusion})$$

$$-\overline{u'_i u'_l}\frac{\partial U_j}{\partial x_l} - \overline{u'_j u'_l}\frac{\partial U_i}{\partial x_l} \qquad (P_{ij} = \textit{production})$$

$$-\left(\frac{g_i}{\rho_r}\overline{u'_j c'} + \frac{g_j}{\rho_r}\overline{u'_i c'}\right) \qquad (G_{ij} = \textit{buoyancy production})$$

$$-c_1\frac{\varepsilon}{k}\left(\overline{u'_i u'_j} - \frac{2}{3}\delta_{ij}k\right) - c_2\left(P_{ij} - \frac{2}{3}\delta_{ij}P\right) - c_3\left(G_{ij} - \frac{2}{3}\delta_{ij}G\right) \qquad (\textit{pressure strain})$$

$$-\frac{2}{3}\varepsilon\delta_{ij} \qquad (\textit{dissipation}) \qquad (5.18)$$

The 3 turbulent flux equations for density reads

$$\frac{\partial \overline{u'_i \rho'}}{\partial t} + U_l \frac{\partial \overline{u'_i \rho'}}{\partial x_l} = \qquad (\textit{local change} + \textit{convective change} =)$$

$$c_{s\rho} \frac{\partial}{\partial x_l}\left(\frac{k}{\varepsilon}\overline{u'_k u'_l}\frac{\partial \overline{u'_i \rho'}}{\partial x_k}\right) \qquad (\textit{diffusion})$$

$$-\overline{u'_i u'_j}\frac{\partial \rho}{\partial x_j} - \overline{u'_j \rho'}\frac{\partial U_i}{\partial x_j} \qquad (P_{ij} = \textit{production})$$

$$-\frac{g_i}{\rho_r}\overline{\rho'^2} \qquad (G_{ij} = \textit{buoyancy production})$$

$$-c_{1\rho}\frac{\varepsilon}{k}\overline{u'_i \rho'} + c_{2\rho}\overline{u'_l \rho'}\frac{\partial U_i}{\partial x_l} - c_{3\rho}\frac{g_i}{\rho_r}\overline{\rho'^2} \qquad (\textit{pressure scrambling}) \qquad (5.19)$$

The turbulent flux equations for concentration are equivalent to the density equations and will not be shown. But an equation for the fluctuations $\overline{\rho'^2}$ is also needed, as introduced by Gibson and Launder (1978):

$$\frac{\partial \overline{\rho'^2}}{\partial t} + U_j \frac{\partial \overline{\rho'^2}}{\partial x_j} = \qquad (\textit{local change} + \textit{convective change} =)$$

$$c_{s\rho} \frac{\partial}{\partial x_j}\left(\frac{k}{\varepsilon}\overline{u'_i u'_l}\frac{\partial \overline{\rho'^2}}{\partial x_l}\right) \qquad (\textit{diffusion})$$

$$-2\overline{u'_j \rho'}\frac{\partial \rho}{\partial x_j} - \frac{1}{R}\frac{\overline{\rho'^2}}{k}\varepsilon \qquad (-\textit{production} \; -\textit{dissipation}) \qquad (5.20)$$

In these equations c_1, c_2, c_3, c_{1p}, c_{2p}, c_{3p}, c_s, c_{sp} and R are empirical constants to be found from experimental results. As discussed by Rodi (1987) the above formulation is not trivial and should not be expected to be universal. Nevertheless, models based on these assumptions have shown significant improvement and generality compared with the first-order-closure models mentioned earlier.

In many environmental flow problems, the fluctuations and their statistical distribution are of highest importance. As seen, the fluctuations are modelled directly in this type of model and on this background a basis for an assessment of extreme values is present. An interesting investigation by Gosman *et al.* (*q.v.*) on this point was presented during the present workshop.

Calculation of turbulent jets on the basis of the Reynolds stress closure has been reported by Malin and Proumen (1989). Calculation of buoyant plumes can be seen by Malin and Younis (1989). The results of the modelling seem to be satisfactory compared withto measurements, especially for the mean properties of the flow. However, the results for the Reynolds stresses and the turbulent fluxes show the right tendency but the modelled values differ by 20 to 40 % from the experimental values. The author is not able to see through these inconsistencies especially because details on the extremely difficult measurements of these correlations are not reported. But it is a little thought-provoking to reflect on the reasons why a model can better describe the mean quantities than the underlying and driving stress quantities.

To end the section on Reynolds stress models it should be mentioned that simplified models (denoted *algebraic stress models*) has been developed, where algebraic equations have been substituted for the transport equations for the Reynolds stresses and fluxes. Significantly faster and less memory consuming computations can be carried out with these types of simplifications without losing the main advantages of the Reynolds stress model; for example, their anisotropic handling of the turbulence near density gradients and boundaries.

5.9 DIRECT SIMULATION - LARGE EDDY SIMULATION

A most interesting approach to turbulence modelling is the large eddy simulation by subgrid-scale modelling, which is not based on the distinction between average velocity and fluctuations. The method in principle models directly as much of the fluctuations in the turbulent energy spectrum as possible in respect to the applied grid. The classic and best known reference to the subject is Smagorinsky (1963). The concept is that, because of the generality of the turbulent kinetic energy spectrum, a grid dependent eddy-viscosity can be determined for the remaining part of the Reynolds stresses, which not are modelled directly. More sophisticated models including transport equations for the individual sub-grid scale stresses and fluxes have since been developed; for a review see Rogallo and Moin (1984).

For 3-dimensional flows the method is of great interest for research and development of other types of turbulence models of above-mentioned types. But in ~~one~~ 2-dimensional depth integrated models theoretical and practical aspects of large horizontal eddies can be studied; for example, the shallow water jets and wakes phenomena presented in this workshop by Jirka (*q.v.*). For practical aspects of large structures in the coastal zone and in estuaries, large eddy simulation will be of high importance.

5.10 DISCUSSION OF THE RELIABILITY OF TURBULENCE MODELS

The author's experiences with turbulence models are limited to the application of k-ε models. But the literature indicates clearly that Reynolds stress models provide an important step toward a higher degree of universality. But, on the other hand, it must also be clearly stated that jets and plumes are probably among the most difficult flow phenomena to model.

Intuitively, an explanation can be found in the general experience that phenomena involving growth always seem difficult to model. In the jet and plume case it is probably the growing length scale which causes problems. Physically, an explanation can perhaps be found in the fact that the above-mentioned turbulence models assume that all changes including the transport of the fluctuations take place without distinction between the eddy sizes. The growing eddies in the jets and plumes cause a delay between the production of the turbulence and the dissipation which is much more pronounced than most other flow phenomena. From a scientific point of view it could be interesting see whether a turbulence model based on a differencing of the fluctuations on wave numbers including a realistic formulation of the energy cascade principle would show improvements.

Meandering of buoyant plumes has caused many difficulties in experimental research. Until now this aspect seems not to be covered by the numerical modelling.

The above-mentioned viewpoints should not be understood as arguments against the practical application of turbulence models. Jets and plumes isolated from ambient flow and turbulence can be modelled better with simple integral models, but when it comes to real flow situations (perhaps including unsteady background flow and density stratification) the turbulence models are simply the only way through. Physical scale models alone cannot be alternatives for practical problems, but laboratory experiments combined with laser-Doppler equipment can be necessary in the adaption and validation of specific aspects of the numerical models.

Finally, it should be mentioned that development of computers will fast increase the importance of the large eddy simulation method. A far-sighted hydrodynamic research strategy should be prepared to take advantage of this potential not only for the results themselves but also because of the inspiration and training in basic thinking the method invokes.

6 Numerical Solutions - Verification and Validation

6.1 CONCEPTS OF VERIFICATION AND VALIDATION

Only a short resume of some basic aspects of the numerical solution techniques can be mentioned in this overview. But the idea is to present concepts of relevance for the discussion of verification and validation of numerical models. As mentioned in the introduction, numerical models are looked upon with a high degree of scepticism from many sides. The reliability of models is therefore essential for their use and recognition in general.

A distinction between verification and validation of models has shown to useful. Here the following definitions of these concepts will be used.

Verification is the way in which the solutions are made probable in respect to the basic mathematical assumptions.

Validation of a numerical model covers a satisfying comparison between measured and modelled results, where the measured and modelled results are coupled only through the common initial and boundary conditions. This way the validation also includes the mathematical formulation of the physics. In a validation the measured results cannot have been used for any calibration of the model.

6.2 GRID TYPES - FINITE DIFFERENCE VERSUS FINITE ELEMENTS

The basic differential equations must be discretized into algebraic equations in order to make them manageable for the computer. Two grid types have developed.

Finite elements. Finite element are the most general numerical solution method, which, in principle, integrate the differential equations over non-overlapping finite volumes of plane surfaces of variable size and shape. One of the advantages of the finite element principle is the fine straight forward adoption to the boundaries. Also the possibility of choosing the optimum grid size around the most interesting areas in the flow domain makes the method an efficient alternative to the finite difference method mentioned below.

The numerical principle in finite elements is the so-called minimum weighted residuals. A general introduction to finite elements is given by Zienkiewics and Taylor (1991), and Hervouet *et al.* (1991) gives an impressive overview over the application of the method in large scale environmental problems.

Finite differences. The most direct and simple discretization is done in a Cartesian grid with equal space increments. However, in almost all applications a non-equidistant grid is preferable in order to obtain the best use of the limited memory of the computer. Probably the most physically-related method is the control-volume method, which is described in detail by Patankar (1980).

The starting point in the control-volume method is the grid for the continuity equation. The flow volume is divided into the desired number of control-volumes for the mass conservation, in which the variables (for example, the density) are located in the cell centre. The velocities in the continuity equation are centred in the surfaces of the cell perpendicular to these. This grid of control-volumes for the continuity equations is easily fitted into the boundary conditions by letting the relevant surfaces follow the boundaries.

Each of the components in the momentum equation is now centred around the velocities already defined. This results in 3 new grids, which are staggered in relation to each other and to the continuity grid. The pressure in the momentum equation will accordingly be centred in the continuity cells.

The control-volume formulation is easy to understand and lends itself to direct physical interpretation. Another aspect, and probably the most important one, is that the resulting solution implies that the integral conservation of volume, mass, momentum, energy etc. is satified exactly over the whole calculation domain and over all sub-domains.

A number of other finite difference methods exists; for example Fletcher (1988) gives a thorough overview. For practical use the introduction of so-called body fitted grids has improved the finite difference methods significantly.

6.3 CONSISTENCY, CONVERGENCE, STABILITY AND NUMERICAL ERRORS

Consistency. The adobted discretization should be consistent with the differential equations, which means that a limit concept, where the increments in space and time vanish, should transfer the finite difference equations back to the differential equations. This is an obvious and basic demand for all finite difference schemes. The analysis will normally be done theoretically by introducing Taylor series expansions for the applied finite differences.

Convergence. Another obvious requirement for the finite difference equations is convergence in the sense that the solution shall converge towards the correct solution when the increments diminish towards zero. Analytical solutions have to be known to verify the convergence of the difference equations.

Stability. Numerical stability in the integration of the governing equations seems to be a obvious necessity for a computer code. For hyperbolic problems (defined in next section) a distinction between explicit and implicit solutions is useful. The explicit solutions normally require restricted increments in time and space whereas the implicit solutions applied in commercial software packages are often so-called unconditional stable. Stable solutions do not only avoid frustrated users from having their calculations broken down many times before a result is given, but implicit solutions are absolutely necessary for professional software which has general purpose. The danger in the application of implicit solutions lies in the fact that a result is always found, even in the cases where the chosen increments in time and space make the difference equations in the model totally unable to describe the original differential equations. The advantage of the explicit models is the strict relation between the physical conditions and the stability criteria.

Numerical errors. As mentioned in section 4, the transport equations are normally most critical in respect to numerical errors. The discretized hydrodynamic equations will normally be analysed in the frequency domain for stability and numerical errors. The so-called Von Neumann principle (Richtmyer and Morton, 1967) is generally accepted.

But the transport/dispersion equation can, because of its different character, be analyzed in the time domain by a simplified approach as described by Vestergaard (1990). Because implicit schemes are difficult or impossible to analyse analytically the method is based on numerical tests. The principle is to introduce an impulse of mass in one grid point and next to run the model in one time step only. The mass distribution in the grid is now analyzed for the moments of the distribution. The moments now characterize the types of numerical errors as follows:

- *Zero order moment.* The zero order moment is the mass balance of the scheme. Mass conservation is an obvious requirement for the numerical scheme. The control-volume approach mentioned always fulfils the requirement of mass conservation.

- *First order moment.* This moment describes the advective velocity of the centre of gravity of the mass in the scheme.

- *Second order moment.* Numerical dispersion, positive or negative, arises from the second order moment. Symmetric schemes in time and space, like the explicit Leap-Frog and the implicit Crank-Nicolson, do not generate numerical dispersion. Unfortunately the Leap-Frog formulation of the transport/dispersion equation is unstable. To supress numerical dispersion some kind of method of characteristics is often applied, where the concentration in the advective term is found by interpolation between the grid points.

- *Third order moment.* The skewness created by the scheme is found from the third order moment. The so-called *wiggles* are often of this type. Fourth-point schemes in space like the *Quick* and *Quickest* (Leonard, 1979; Bascco, 1984) are able to suppress wiggles and are applied in turbulence models. Often wiggles are absorbed by the physical or numerical dispersion in the scheme after a few time steps.

- *Four and higher order moments.* Symmetrical oscillations or wriggles can be seen as fourth order moment errors also known as flatness. Again these wriggles often quickly disappear due to the dispersion. Although it is obvious from analyses based on Taylor series expansion that the lower order errors normally are the most significant, the risk always exists that higher order errors can show up, for example, in strongly geometrically distorted schemes.

The choice of increments in time and space is essential in numerical modelling. Even the most advanced mathematical formulation will fail if the choice is wrong. For implicit models based on the coupling of several staggered equations only a number of numerical tests can make a satisfying verification of the set up. According to the concepts of consistence and convergence, tests where the solution is proved to be invariant of the chosen increments should always be performed. This can only be done in tests where the increments are reduced step by step. Also, tests where the grid direction varies compared the the main flow direction should be done.

6.4 PARABOLIC, ELLIPTICAL OR HYPERBOLIC SOLUTIONS

The mathematical terms parabolic, elliptic and hyperbolic are commonly used for the classification of differential equations. Patankar (1980) and others have defined a parallel terminology for the numerical solution techniques. A parabolic solution is a solution where the influence travels only in one direction. An example is the integral model of the submerged buoyant jet, where the calculation starts at the outlet and propagates along the centerline. The general Navier-Stokes equations for incompressible flow have an elliptic solution because influence travels in both, or rather, in all directions. A change in one point will immediately propagate to all other grid points. But when a compression term is included in the continuity equation the flow will become compressible and now the solution becomes hyperbolic.

Elliptic solutions are in principle the most time-consuming because all equations in all grid points must be satisfied simultaneously. The most well-known procedure is the SIMPLE method (Patankar, 1980), which includes the above mentioned control-volume discretization. The method is based on iteration and in practice the full Navier-Stokes equations require a large number of iterations on each time step. For steady state problems with a unidirectional flow, a considerable increase in calculation speed can be achieved if a parabolic solution is acceptable.

To speed up the calculation, especially for transient elliptic problems, some codes transfer the problem to a hyperbolic one by adding an artificial compressibility in the equation. This makes it possible to calculate the continuity and the momentum equations staggered in time so the iteration can be avoided. Often the compressibility chosen is without any physical relevance, aside from the rare cases in civil engineering where the Mach-number is critical.

7 Examples of Civil Engineering Applications

7.1 GENERAL

In civil engineering applications jets and plumes are most often only subproblems in a larger context. Discharge of sewage from long sea outfalls and discharge of cooling water from power production are well-known examples. But the establishment of bridges and tunnels including temporary structures also often contain hydraulic aspects of jets and plumes. Especially the environmental problems are usually of high complexity.

The complexity can often be related to a very wide span in time and length scales. In many water quality problems the discharge itself take place in length scales of 1 m as an order of magnitude. But the consequence of, for example, bacterial contamination or eutrofication, will often appear on a horizontal length scale of 10 to 100 km. Bearing in mind that the actual and practical maximum number of grid points in numerical computations lies in the order of magnitude of 10^5 to 10^6 this leads to the conclusion that one model seldom can cover the overall problem.

Another, perhaps even more important degree of complexity, is the coupling of a variety of physical, chemical and biological processes into one integrated numerical model. For reasons of completeness it should be mentioned that the question of uncertainty in some of the processes often leads to the introduction of propabalistic elements in the models; an issue which will not be discussed further here.

To illustrate these two forms of complexity, two examples are given below. First, the modelling of the bathing water quality of the Barcelona long sea outfall can be characterized to be complex in time and geometric scales. Secondly, the modeling of the effluent quality of sedimentation tanks and clarifiers is complex in respect to the numerous processes involved.

7.2 BARCELONA LONG SEA OUTFALL

The development and centralization of the sewage treatment in cities has resulted in an increase in direct discharges of sewage to the sea. Long sea outfalls are now the final part of many cities' sewer systems.

The numerical modelling of the expected pollutant field from the Barcelona long sea outfall was carried out by *HR Wallingford* and are described in detail by Mead and Cooper (1992). A resume of the strategy and the basic ideas is presented here.

The hydrodynamic model was based on the Navier-Stokes equations, assuming hydrostatic pressure in the vertical direction. The finite difference equations were solved explicitly in the horizontal plan and implicitly in the vertical. A zero-equation turbulence description based on the mixing length concept was used, including empirical suppression of vertical mixing. To this model a transport model for temperature and salinity was coupled. The model ran on a AMT DAP parallel processing computer, with 4096 processors. The simulation covered a period of 2 months, which is the critical bathing season.

The following models were applied:

- The regional model. This hydrodynamic model covered an area of 80 km along the coast and 30 km off-shore. The horizontal grid size was 1 km and the vertical grid was 30 m. The boundary conditions were formulated from measurements of current, temperature, salinity and wind.
- The local model. The local hydrodynamic model used a horizontal grid size of 150 m and a vertical grid size of 5m. The boundary conditions were taken from the regional model.
- Buoyant plume model. To calculate the initial dilution and the plume trapping level an integral plume model was used. This model was the EPA model (US EPA, 1985). The ambient current and density fields were taken from the local model.
- The bacterial dispersion model. The bathing water quality expressed as concentration of E.coli bacteria was modelled with a 3-dimensional random walk model. The flow field and the dispersion coefficients were taken from the local model and the source concentrations and trap levels were taken from the buoyant plume model.

7.3 JETS AND PLUMES IN SEDIMENTATION TANKS AND CLARIFIERS

Sedimentation tanks also known as clarifiers form the final stage in treatment plants. The inflow to the clarifiers normally has a concentration of an order of magnitude of 5 kg/m^3 suspended matter equal to a density of around 0.5 % above density of the water itself. Effluent standards often require concentrations lower than 0.02 to 0.1 kg/m^3. The basic principle of the removal of the suspended matter is sedimentation by gravity.

Modern clarifiers are rectangular or circular reinforced concrete structures of a horizontal size of 30 to 60 m and with a depth of 2 to 5 m. Close to the bottom, mechanical devices like scrapers operate to remove the sedimented sludge to a deeper ditch from where the sludge is pumped away.

The clarifier can be characterized by the following 3 zones:

- The inlet zone where the inlet near the surface forms a jet with negative buoyancy. This jet or plume falls towards the bottom and will to some degree erode and re-suspend the sedimented sludge here. An important effect in this zone is the flocculation of the particles. The degree of flocculation has significant influence on the settling characteristics. The transport of suspended matter is accordingly modelled in a multi-fractional description from which the settling can be derived as function of turbulence and concentration.

- The sedimentation zone where a sludge blanket is formed. Above the sludge blanket the average flow moves towards the outlet. Below the sludge blanket a density current likewise moves towards the outlet but simultaneously a sedimention to the lower non flowing zone takes place. The flow here is stratified, and undesirable mixing across the sludge blanket can occur when the shear stress increases. Below the sludge blanket the process can be characterized as hindered settling, where the displacement of the water by the particles reduces the natural settling. The high concentration of particles more or less damps out the turbulence here, and in the lower part the rheological properties of the sludge, modelled by a Bingham yield strength, have significant influence.

- The outlet opposite the inlet also occurs near the surface to avoid the sludge being withdrawn by the Bernouilli effect from the sludge blanket. The problem is parallel to the so-called selective withdrawal phenomenon in a water reservoir.

The inlet zone where the falling jet/plume occurs is the main source of turbulence in the tank. The geometric and hydraulic design of this area is substantial for the overall performance of the clarifier.

The first consistent attempt to set up a numerical model for this complex of flow and particle dynamics was presented by Larsen (1977). Based on those valuable experiences the modelling has been taken up again at Aalborg University in co-operation with I. Krüger AS, Copenhagen. This work will be reported in late 1993, but the principles are outlined by Dahl *et al.*(1991).

8 Conclusions

From the above brief discussion and description of numerical modelling of jets and plumes in civil engineering some main points can be mentioned:

- A wide spectrum of numerical models for jets and plumes are available from the simple one-dimensional integral models running in a spreadsheet program to the advanced 3-dimensional turbulence model based on second order turbulent closure formulation and running on DAP (distributed array processors) or vector computers.

- For specific problems the simple integral models are best verified with experimental results. But these models cannot describe the influence of the ambient turbulence and the transition from the jet and plume stage to the passive dispersion stage.

- Simple turbulence models like the zero and one-equation models are not adequate for jet and plume modelling. The k-ε-models can be applied with some modification, which limits the general use.
- Advanced turbulence models like the Reynolds stress models, which are able to handle the anisotrophy of the turbulence and the influence of buoyancy more correctly, seem to give confident results. Nevertheless, the author would welcome a detailed experimental validation of these models including a validation of each term in the turbulence formulation.
- Direct simulation or large eddy simulation seems to be a promising supplement to experimental work in the research of jet and plumes. For jets in shallow waters, where depth-integrated models are appropriate, direct simulation could be applied in practise.
- Because of the increasing importance of numerical models, the author's viewpoint is that further research on jets and plumes should be carried out within a combination of experimental and numerical work.

These points are important from the author's point of view in respect to context of the present workshop, the objectives of which are to be more reflective than specific. The author hopes that the reader agrees as well as disagrees in various points in this attempt to give an overview on the subject.

9 Acknowledgements

The author is grateful to *HR Wallingford* and the *Danish Hydraulic Institute* for providing the information on the modelling of the Barcelona long sea outfall and modelling of the surface plumes respectively. Also, the author should like to thank Ole Petersen for useful comments.

REFERENCES

ASCE, 1988, Turbulence Modelling of surface water flow and transport, Part I-V, ASCE Task Committee on Turbulence in Hydraulic Computations. *J. Hydraul. Engng*, ASCE, **114**, No 9, 970-1074.

BASCO, D.R., 1984, *An implicit, wiggle-free, and accurate upstream algorithm for the one-dimensional transport diffusion equation.* Report EL-84-4, U.S. Army Eng. Waterways Experiment Station, Vicksburg, USA.

CHEN, C.J. and RODI, W., 1975, A mathematical model for stratified turbulent flows and its application to buoyant jets, *Proc. 16th IAHR Congress*, Sao Paulo.

DAHL, C.P., LARSEN T. and PETERSEN O., 1991, Development of a numerical model for secondary clarifiers, *Proc. 25th IAHR Congress,* (ed. A. Nieto) C-161, Madrid.

DALY, J.B. and HARLOW, F.H., 1970, Transport equations in turbulence. *Phys. Fluids,* **13**, No. 11, 2634-2649.

FAN, L.-N., 1967, Turbulent buoyant jets into stratified or flowing ambient fluids, *Rep. KH-R-15*, W.M. Keck Lab., Calif. Inst. Tech., USA.

FAN, L.-N. and BROOKS N.H., 1969 Numerical solutions of turbulent buoyant jet problems, *Rep. KH-R-18*, W.M. Keck Lab., Calif. Inst. Tech., USA.

FLETCHER, C.A.J., 1988, *Computational Techniques for Fluid Dynamics*, Vol. I and II. Springer Series in Computational Physics. Berlin.

GIBSON, M.M. and LAUNDER B.E., 1978, Ground effects on pressure fluctuations in the atmospheric boundary layer, *J. Fluid Mech.*, **86**, 491-511.

GOSMAN, A.D., LIU, R. and MCGUIRK, J.J., 1994, Prediction of mean and fluctuating scalar fields in buoyant jet with crossflow problems, this volume.

HANSEN I.S., 1991, Transport of pollutants in the near field, part I and II. *Course Notes from Short Course on Computer Modelling in Ocean Engineering*. Stiges, 26 - 28 Sep. 1991.

HERVOUT, J.M., LABADIE G. and LEPEINTRE F., 1991, Progress in the development of numerical tools for environmental studies. *in Environmental Hydraulics*, (eds J. H. W. Lee and Y. K. Cheung), 713-728, Balkema, Rotterdam.

HIRST, E., 1971, Bouyant jets discharged into quiescent stratified ambients, *J. Geophys. Res.*, **76**, 30.

JIRKA, G.H., 1994, Shallow jets, this Volume.

LARSEN, P., 1977, On the hydraulics of rectangular settling basins. *Report No. 1001*, Dept. of Water Resources Eng. University of Lund, Sweden.

LARSEN, T., 1983, Sea outfall design based on a stochastic transport/dispersion model, *in Modeling of Environmental Flow Systems,* (eds R. A. Bajura and T.B. Morrow), 19-24, American Society of Mechanical Engineers, New York.

LARSEN, T., PETERSEN, O. and CHEN, H.B., 1990, Numerical experiments on turbulent buoyant jets in flowing ambients, *Proc. 8th Int. Conference Computational Methods in Water Resources*, Venice, June 1990.

LAUNDER, B.E. and SPALDING D.B., 1974, The numerical computation of turbulent flow, *Comp. Meth. In Applied Mech. Eng.*, **3**, 269.

LEE, J.H-W. and CHEUNG, 1990, Generalized Lagrangian Model for Buoyant Jets in Current, *J. Environm. Engng*, ASCE, **116**, NO.6, 1085-1106.

LEONARD, B.P., 1979, A stable and accurate convection modelling procedure based on quadratic upstream interpolation. *Computer Methods in Applied Mechanics*, **19**, 1.

MALIN, M.R. and PROUMEN, 1989, Calculation of intermittent turbulent shear flow with a Reynolds stress transport closure. *Proc. 3rd International PHOENICS User Conference.* (eds CHAM Ltd. London), 410-424.

MALIN, M.R. and YOUNIS B.A., 1989, Calculation of turbulent buoyant plumes with a Reynolds stress and heat flux transport closure. *Proc. 3rd International PHOENICS User Conference.* (eds CHAM Ltd. London), 368-390.

MEAD, C.T. and COOPER A.J., 1992, Three-Dimensional numerical modelling of coastal hydrodynamics and pollutant dispersal. *Proc. 2nd International Conference on Hydraulic and Environmental Modelling of Coastal, Estuarine and River Waters*, (eds R.A. Falconer, S.N. Chandler-Wilde and S.Q. Liu), 31-41, Asgate Publ., U.K.

MØLLER J.S., SCRØDER H. and HANSEN I.S., 1990, An integral model of the buoyant jet and plume in a cross current and wind, *in Stratified Flow*, (eds E.J. List and G.H. Jirka), 822-835, ASCE, New York.

MORTON, B., TAYLOR G.I. and TURNER J.S., 1956, Turbulent gravitational convection maintained and instantaneous sources, *Proc. R. Soc. Lond.*, Ser. A, **234**, 1-23.

PATANKAR, S.V., BASU, D.K. and APAY, S.A., 1977, Prediction of the three-dimensional velocity field of a deflected turbulent jet. *J. Fluids Engng.*, Trans ASME, Dec 1977, 758-762.

PATANKAR, S.V., 1980, *Numerical Heat Transfer and Fluid Flow,* Hemisphere Publishing Corporation.
PETERSEN, O., 1993, Laboratory and numerical experiments on the dilution of buoyant surface plumes, this Volume.
PETERSEN, O. and LARSEN T., 1989, Infrared sensing of buoyant surface plumes. *Proc. 3rd. Int. Symposium on Refined Modelling and Turbulence Measurements,* IAHR, Tokyo.
PETERSEN, O. and LARSEN T., 1990, Laboratory study of dispersion of buoyant surface plumes. *Proc. Int. Conference on Physical Modelling of Transport and Dispersion*, (eds E. Adams and G.E. Hecker), ASCE, MIT Mass., USA.
PETERSEN O., 1992, Application of turbulence models for transport of dissolved pollutants and particles, *Ph.D. Thesis, Department of Civil Engineering, Aalborg University, Denmark.*
RICHTMYER,R.D. and MORTON, K. W., 1967, *Difference Methods for Initial Values Problems*, Interscience Publishers, New York.
RODI, W., 1984, *Turbulent models and their applications in hydraulics*, State-of-the-art review, IAHR, Delft, The Netherlands.
RODI, W., 1987, Examples of calculation methods for flow and mixing in stratified fluids, *J. Geophys. Res.*, **92**, C5, 5305-5328.
ROGALLO, R.S. and MOIN P., 1984, Numerical simulation of turbulent flows, *Ann. Rev. Fluid Mech.*, **16**, 99-137.
SCHLICTHING, H., 1969, *Boundary Layer Theory*, McGraw Hill, New York.
SMAGORINSKY, J., 1963, General circulation experiments with the primitive equations. part I. The basic experiment. *Mon. Weather Rev.* **91**, 99-164.
US EPA, 1985, *Initial mixing characteristics of municipal ocean discharges: volume 1, procedures and applications.* Newport, Oregon
VESTERGAARD, K., 1991, Numerical errors in transport/dispersion modelling, *in Environmental Hydraulics* , (eds J. H. W. Lee and Y. K. Cheung), 1245-1250, Balkema, Rotterdam.
WEIL, J. and FISCHER, H.B., 1974, Effect of stream turbulence on heated water plumes, *J. Hydraulic Div.*, ASCE, **100**, HY7, 951-970.
WOOD, I.R., 1993, Asymptotic solutions and behaviour of outfall plumes, *J. Hydraul. Engng.*, ASCE, **119**, 5, 555-580.
ZIENKIEWICS, O.C. and TAYLOR, 1991, *The Finite Element Method*, McGraw-Hill, New York.