NUMERICAL MODELS IN COOLING WATER CIRCULATION STUDIES: TECHNIQUES, PRINCIPLE ERRORS, PRACTICAL APPLICATIONS

G. S. Rodenhuis
Danish Hydraulic Institute
DK-2970 Hørsholm, Denmark

ABSTRACT

Four principle models used in cooling water circulation studies are introduced. The coupling problems arising when information has to be transferred from one model to another are discussed and sources of possible errors identified. The errors introduced when the various equations involved are solved, are described together with possible techniques to avoid such errors. The paper demonstrates that no fail-safe methods are available and suggests that results are used only with full awareness of the possible errors.

INTRODUCTION

In a conference on waste heat management the objectives of the modeling work in connection with the design of a large thermal plant need no explanation. In this paper we shall look at the problems that arise and the possible errors that may occur when numerical models are used for predictions of the distribution of temperatures around the point of outfall and for the computation of the transport and dispersion of the waste heat in the receiving water body. Estimates of the possibility of recirculation and of the ecological impact are based on such models. The point that we wish to make is that results of such models should be used with a full awareness of problems and sources of possible errors.

Before going into these aspects of numerical modeling it may be relevant to set-off numerical modeling against physical modeling. Why is it that the modeling work is mainly done with the use of numerical models? The answer lies in the fact that dispersion of heat and transfer to the atmosphere are difficult to model and this is again related to modeling of the different scales of turbulence. Abraham - Ref.[1] - demonstrates in fact that turbulent stresses and transports can be modeled correctly, but only for conditions with a high Reynolds number and where

Figures referred in the text are given at the end of the paper.

Preceding page blank

a critical value for this number can be established as a solid experimental fact. Jet flow is a problem where this criterion applies.

For computations of the transport and dispersion of heat away from the outlet - the far-field - mathematical models have to be used. The fact that often irregular flow patterns and topographies have to be described means that mathematical models usually are numerical models rather than analytical. However, here the problem of correctly representing the physical processes is hardly smaller. This problem is principally related to lack of detail in the description in the models. The equations describing the instantaneous movements of particles in three dimensions are intractable. In order to have a "workable" model we have to use time and space averaged forms and this introduces dispersion terms. Fickian-type formulations appear to give a workable description. However, the wide variety of time and length scales involved makes a unified approach with universally applicable expressions for the dispersion coefficient in these formulations impossible. The correctness of the description of the physical processes in mathematical form is discussed extensively in the literature. Although we may here have a first source of errors, we do not intend to discuss these in this paper.

In order to arrive at "workable" models it is furthermore necessary to divide the region in which the processes of dilution, transport, dispersion and transfer to the atmosphere take place into a near-field around the discharge and a far-field, with different models for the processes in these fields. This introduces the problem of coupling these various models and here errors can be introduced. The models that we shall consider in this connection are a plume model for the near-field and hydrodynamic (HD) and transport-dispersion (TD) models for the far-field. The plume model we have in mind has the form of a Gaussian distribution of velocity and excess temperature around a centerline value. The hydrodynamic model in this connection is usually an expression of the vertically integrated equations of conservation of mass and momentum over one or two horizontal dimensions. Associated with this is a one or two dimensional TD-model. For later reference we give here the form for two-dimensions

$$\frac{\partial c}{\partial t} + u\frac{\partial c}{\partial x} + v\frac{\partial c}{\partial y} = \frac{1}{h}\frac{\partial}{\partial x}\left(hD_x\frac{\partial c}{\partial x}\right) + \frac{1}{h}\frac{\partial}{\partial y}\left(hD_y\frac{\partial c}{\partial y}\right) - \frac{\alpha c}{h} + \Sigma\frac{Q_L(c_L-c)}{h\,\Delta x\,\Delta y} \quad (1.1)$$

where the notation is

c - excess temperature
c_L - excess temperature discharged
u,v - horizontal velocity components, integrated over depth,

in resp. x- and y-directions
h - water depth
D_x, D_y - dispersion coefficients in resp. x- and y-directions
Q_L - discharge of outlet
α - first order decay factor for heat.

In order to assess ecological impact the three hydraulic models may be supplemented with an ecological mathematical model.

As will be discussed, coupling problems exist between all four models. Apart from errors arising from such problems the numerical solution of the actual equations may also introduce errors. These are usually related to a necessary discretisation of the domain of the models.

Before continuing with a discussion of possible errors it must be remarked that it lies in the nature of the problem that computed results even when including considerable errors usually look "reasonable". The TD-model seldom becomes unstable or otherwise indicates its incorrectness. It is difficult to judge from a plot of temperature contour lines whether the results are correct or incorrect.

COUPLING PROBLEMS

Coupling of Near-Field and Far-Field Models

At the outfall of a thermal power plant a certain mass of water with a certain amount of waste heat is discharged into the receiving water body with a certain amount of momentum. Using a plume model, the distribution of these quantities around the outfall can be modeled. The effect of introducing mass and momentum should be represented in the HD-model that is used to describe the far-field hydrodynamics and the excess temperature distribution around the outfall must be transferred to the TD-model. However, the resolution in HD and TD-models is usually quite different than that used in plume models. In fact most HD and TD-models have a discrete representation, whereas plume models usually have a continuous representation. In the process of transferring quantities computed in the plume model to the other models, errors are introduced.

Coupling Plume and HD-Model

Buoyancy and remaining jet momentum induce horisontal velocities in the receiving water body. If a high velocity surface jet is utilized the jet momentum can induce a current pattern which can be of considerable importance for the shape of the entire temperature field, especially in situations where the currents in the receiving water body are weak. One should consider here that the instantaneous shape of the jet usually is a meandering plume. Averaged over a certain time the jet appears as a much wider 'steady-state' plume. Clearly these effects are difficult to represent in, for example, a finite difference HD-model where the grid size must be in the order of hundreds of meters for reasons of computational economy.

Coupling HD and TD-model

The mass of the discharged water can be represented in the HD-model as lumped over a few meshes of the grid and this procedure needs not introduce errors. However, an error can be introduced by incorrect coupling of the equation of mass of the HD-model with the equations of the TD-model. The error can be easily avoided by following a correct modeling procedure. The point is, however, that when this error is made it is not easily detected. Another error, that of numerical dispersion, may mask it.

A HD-model is usually used to create the flow distribution in the receiving water body. Neglecting the contribution of the source in the mass-equation in this model will hardly be noticed in the velocity field obtained as the more important contribution of momentum is also - out of necessity - neglected. Observing little effect in the HD results may tempt one to neglect the source term in the mass equation when deriving the equations for the TD-model. We may illustrate this below.

In deriving the TD-equation (1.1) given in the previous section from the principle equation

$$\frac{\partial(hc)}{\partial t} + \mathrm{div}(h\,\overline{V}\,c) - \mathrm{div}(h\,\overline{\overline{D}}\,\mathrm{grad}\,c) + \alpha c = \Sigma Q_L\, c_L \qquad (2.1)$$

, the hydrodynamic mass equation

$$\frac{\partial h}{\partial t} + \mathrm{div}(h\,\overline{V}) = \Sigma Q_L \qquad (2.2)$$

, should be used. The source term resulting in eq. 1.1 has the form

$$\Sigma\ \frac{Q_L(c_L-c)}{h\ \Delta x\ \Delta y} \qquad (2.3)$$

However, if the contribution of the source is neglected in eq. 2.2 the source term will be

$$\Sigma\ \frac{Q_L\ c_L}{h\ \Delta x\ \Delta y} \qquad (2.4)$$

The notation here is that of the previous section with further

$\bar{V}$, the vector (u,v)

$\bar{\bar{D}}$, the matrix $\begin{bmatrix} D_x & o \\ o & D_y \end{bmatrix}$

The effect on the temperature distribution around the source is dramatic; the temperature will in fact rise with each following step in time in the computations. The effect is shown in Figs. 1.a, b and c, where results are presented obtained in connection with a site selection study made for a combined desalination and power plant on the east coast of Saudi Arabia. The discharge of this plant will be 176 m^3/s with an excess temperature of 7°C. Fig. 1.a shows the most correct solution in which the mass term is included in the equation and where no numerical dispersion is present. Fig. 1.b shows the same situation, now neglecting the mass term, but still without numerical dispersion: high temperature around the outlet causes the 1°C contour to stretch over a much larger area. Fig. 1.c shows the same situation now neglecting the mass term and with numerical dispersion. One observes that both errors almost cancel each other: the results resemble those of Fig. 1.a. So even with two considerable errors the results may still look reasonable! In Fig. 1.d results from a hydrodynamic model are shown for the strait between Saudi Arabia and the island of Bahrain. The velocity field for the TD-model is taken from this HD-model.

Coupling Plume and TD-model

The mass error and, as we shall see, also numerical dispersion can be avoided. Transferring the temperature distribution from the plume model to the TD-model presents a more principle problem. The usual approach is to assume that waste heat is distributed over a few grid spaces and uniformly mixed from bottom to surface. The integral value of excess temperature times volume, taken per unit time, is set equal to the waste heat discharged, but different volume, excess temperature products may be taken to correspond to the same amount of waste heat. It is difficult to make this distribution equivalent to that obtained in the plume model where the distribution is principly three-dimensional. Using a different number of grid spaces for the near field distribution gives different far-field contour lines. The difference is greatest close to the source and becomes less further away. However, with a very different near-field distribution the picture of the contour lines will be very different. An extreme example is given in Figs. 2.a and b. The results shown are obtained in connection with the site investigation for a 4000 MW nuclear plant in Denmark. Fig. 2.a shows the contour picture obtained with a near-field distribution over eight grid spacings, whereas in Fig. 2.b the distribution is over only one grid spacing. Of course the very high excess temperature in the last case will lead to a high rate of heat loss to the atmosphere and the contours cover a smaller area. Again numerical dispersion could mask the error caused by an incorrect near-field distribution.

The choice of near-field distribution in the TD-model must of course be guided by the results obtained in the plume model, but because of the different character of the two models the transfer will always be imperfect.

The problem outlined above is, however, not the only principle problem in coupling a plume model to a TD-model. The plume model assumes a certain temperature of the ambient water. Characteristics of the plume such as entrainement and buoyancy are dependent on this ambient temperature. One should realise in this connection that the 'far-field' is also present near the outlet. This is especially the case for a coastal outlet with an oscillating tidal current along the coast. In the case of a power plant with a large waste heat discharge the temperature in the area around the outlet will build up and the entraining water will already be heated. This leads us to a circular problem in modeling: in order to compute the near-field temperature distribution by means of a plume model the ambient temperature must be known, but in order to compute the ambient temperature one must compute the far-field distribution and this is in turn dependent on the near field distribution given by the plume. One is forced to some sort of iterative procedure if accurate answers are to be obtained.

After having looked at these problems of coupling we can conclude that our difficulties stem principally from handling the problem in separate models and from differences in resolution of these models. This approach is imposed by engineering necessity. The problem of difference in resolution may be reduced by using a model with varying resolution: detailed around the outlet, with a coarser grid away from the outlet. A Finite Element Model allows a net with small elements around the outlet gradually changing to larger elements further away. In a Finite Difference Model a change-of-scale can be used, using a local patch of high resolution in an otherwise coarse grid.

The circular problem of the value of the ambient temperature to be used in the plume computation is hard to overcome. One is in fact looking for a "complete" model, where the near-field and far-field temperatures are computed simultaneously. Attempts have been made to develop such a model using the Marker-in-Cell technique. The entire flow region in three dimensions is divided into a sufficient number of cells and the computation procedure is based on the approximate satisfaction of the integral form of the conservation equations for each cell at every time step; the approach is conceptually that of "box" modeling. The demand on computer storage and time is, however, excessive and for engineering applications not acceptable.

Coupling with Ecological Models

Temperature is an important forcing function in biological and chemical processes. Ecological models can be applied to compute the consequences of discharges of waste heat. However, such models usually deal with slowly varying processes with a time scale expressed in weeks and months. The information required has for example a form as "number of days per year for which a certain temperature level is exceeded". There is a clear conflict of time scales between the models for temperature distribution and the ecological models. The approach usually is to simulate in the temperature distribution models a short period that is statistically equivalent to the period required in the ecological model. This last period can be, for example, a year. This introduces a statistical interface between the temperature distribution models and the ecological model. Errors will be introduced, if the quality or quantity of the data on which the statistics are to be based, is insufficient.

There is also a conflict in the spatial scales between the HD- and TD-models and an ecological model. In a complex ecological model a large number of ordinary differential equations have to be solved for each mesh considered. If a grid of the same mesh size as used in the HD- and TD-models would be used in order to obtain detailed information on fluxes, the cost of computation would become prohibitive. An averaging of hydraulic conditions must be introduced with, as a consequence, a loss of accuracy.

For a more detailed discussion of the use of ecological models in power plant site studies we refer to the paper by K.I. Dahl Madsen - Ref.[2]- in this Conference.

NUMERICAL TECHNIQUES AND ERRORS

In this section we shall limit the discussion to errors in the TD-models and techniques to avoid these. Not that the other models have no errors, but particularly in the TD-model the errors are difficult to detect and the results may give a false impression of correctness. Moreover, the TD-model has a central rôle in waste heat studies.

When the continuous differential equation of transport and dispersion is represented on a discrete grid errors can be introduced. Well-known is the numerical dispersion that may be introduced. But although well-known it is difficult to avoid without introducing other errors. We may in short recall how this error is introduced. A simple finite difference approximation of the terms

$$\left[\frac{\partial c}{\partial t}\right]_{n,j}^{n+1} + \left[u\,\frac{\partial c}{\partial x}\right]_{n}^{n} \quad - - - - \tag{3.1}$$

to $$(c_j^{n+1} - c_j^{n})/\Delta t + u(c_j^{n} - c_{j-1}^{n})/\Delta x \tag{3.2}$$

introduces the truncation error

$$u(\Delta x - u\Delta t)\,\frac{\partial^2 c}{\partial x^2} \quad - - - - \tag{3.3}$$

, which has the form of a dispersion term. The term depends on the choice of the grid and the magnitude of the advective velocity. Its value may be many times that of the physical dispersion and may completely invalidate the results.

In order to avoid this error one may resort to higher order difference approximations. However, then a residual numerical radiation and skewness appear. The various effects are illustrated in Fig. 3.

Numerical oscillations also may be introduced by use of an inappropriate scheme. For example, the transport equation

$$\frac{\partial c}{\partial t} + u\frac{\partial c}{\partial x} = 0 \tag{3.4}$$

may be approximated by the centred, second order difference scheme

$$\frac{1}{2}\left(\frac{c_j^{n+1} - c_j^{n}}{\Delta t} + \frac{c_{j-1}^{n+1} - c_{j-1}^{n}}{\Delta t}\right) + \frac{u}{2}\left(\frac{c_j^{n+1} - c_{j-1}^{n+1}}{\Delta x} + \frac{c_j^{n} - c_{j-1}^{n}}{\Delta x}\right) = 0 \tag{3.5}$$

However, when initially $c_j^0 = 0$ for all j and u is constant, this provides

$$c_j^1 = -\left(\frac{1 - u\frac{\Delta t}{\Delta x}}{1 + u\frac{\Delta t}{\Delta x}}\right)c_{j-1}^1 \tag{3.6}$$

$$= (-1)^j\left(\frac{1 - u\frac{\Delta t}{\Delta x}}{1 + u\frac{\Delta t}{\Delta x}}\right)^j c_0^1 \tag{3.7}$$

which oscillates for all Courant numbers $Cr = u\Delta t/\Delta x$, satisfying $|Cr|<1$.

Higher order approximations have the disadvantage that they extend over more and more grid points with increasing orders of approximation. This presents problems at the boundaries. Artificial assumptions for the approximation have to be made and errors are introduced.

One method to avoid all such types of errors is to apply a Lagrangian type of model. The advective term is then in fact cut out by moving in a local frame with the local velocity at each grid point. When this procedure is followed in a flow field varying in time and space, the grid will distort to unwieldy forms. Therefore the local frame is moved only for one time step and the information obtained is then transferred back to points in a fixed grid. One may also express it in a complimentary form: given a fixed grid at time $t_o + \Delta t$ where was the information now in grid point B at time t_o. The principle is illustrated in Fig. 4. Following this principle a practical model has been developed at the author's Institute.

The method requires a sophisticated interpolation technique to determine the values of concentration (or temperature) in the points at time t_o, relative to the given fixed grid. The interpolation used is based on a 12-point Everett interpolation. The "correctness" of this approach is best illustrated in a so-called rotation test in which a Gaussian distribution is rotated in a two-dimensional grid around a center point located outside the distribution. Results are presented in Fig. 5. It may be observed that the shape is fairly well preserved in this rather tough test.

This type of approach may also be used with a Finite Element Model. As the F.E.M. technique does not give solutions in discrete grid points, but as solution surfaces over elements, the interpolation is not required (- or is in fact already included in the F.E. M. technique). Clearly higher order elements are necessary to obtain results without erroneous dispersion. However, compared with finite difference schemes F.E.M. models are usually found to be considerably more expensive for time varying solutions.

The 12-point scheme introduced above also requires special formulations at the boundaries, but satisfactory approximations can be obtained. Simply using the correct zero concentration value for points beyond a land boundary gives good results. This is demonstrated in a so-called L-test in Fig. 6.a.

The propagation of a circular distribution around a sharp corner is depicted for a sequence of time steps. One observes that the circular form is preserved for the stretch before the corner. It is distorted beyond the corner, but this is physically realistic. The deformation is caused by the flow field around the corner. In order to illustrate how an incorrect scheme with considerable numerical dispersion may distort a circular distribution, corresponding results of such a scheme are shown in Fig. 6.b.

At open water boundaries also special attention is required. However, satisfactory results are usually obtained when such boundaries are recognized as being either inflow or outflow boundaries, with an assumption on the mixing conditions in the water body beyond the boundary.

A quite different method, that probably is the most accurate, is based on a spectral technique. The method is developed for an application in atmospheric pollution by Christensen and Pram - Ref. 4. The technique was applied to hydraulic problems in the author's institute. The method is very accurate, but computer costs are about four times as high as for the 12-point scheme. The method also has limitations with regard to resolution of realistic topographies. In short the technique is as follows: It is assumed that c can be approximated by a finite Fourier representation

$$f(x,t) = \sum_{k}^{K} A(k,t)\, e^{ikx}, \text{ with } i = \sqrt{-1} \qquad (3.8)$$

For a given continuous function f(x,t) one can always find A(k,t) such that f(x,t) = c(x,t) on grid points $x_j = j\Delta x$:

$$A(k,t) = \frac{1}{J} \sum_{j}^{J} c_j(t)\, e^{-ik\, j\Delta x} \qquad (3.9)$$

The simple equation

$$\frac{\partial c}{\partial t} + u\frac{\partial c}{\partial x} = 0 \qquad .10)$$

transforms to

$$\frac{\partial A(k,t)}{\partial t} + u\, i\, k\, A(k,t) = 0 \qquad (3.11)$$

From this equation A(k,t) can be computed at each new time $n\Delta t$ with A(k,t) given, c can be obtained from an inverse Fourier-transform. The point here is to note that the advection term in (3.11) is not approximated by a finite difference form so that numerical dispersion is avoided.

DISCUSSION

A discussion of modeling problems and errors may leave the reader with a rather gloomy impression of the whole modeling effort. And there is more, in addition to the problems discussed there are the difficulties encountered when dispersion coefficient and heat transfer coefficients have to be selected. Field investigations before the plant has been constructed can only give a very limited impression of the mixing characteristics of the receiving water body, as discharge volume, discharge momentum and buoyancy cannot be represented.

However, if the models are used with an awareness of the inaccuracies and if sensitivity tests are made for the important parameters, usefull predictions can be made. Such predictions would allow design considerations of outlet-intake construction cost against possible recirculation as discussed in the paper by Schrøder - Ref. [5] - in this Conference, and an evaluation of ecological consequences.

We may underline the above statement with a final result. The possibility of simulating the transport-dispersion process over an extended period of time in a realistic topography is illustrated in Fig. 7. The transport and dispersion of a conservative substance with an irregular release - 16 hrs out of 24 per day - is simulated. The area concerned is Køge Bay, south of Copenhagen. In a sequence of plots the development of the concentration field from an initial distribution to the situation after one week is shown. The results after one week are compared to measurements. One may observe that after having been through considerable changes, the characteristics of the field after seven days compare well with measurements.

ACKNOWLEDGEMENT

The present paper is based on research by the staff of the Computational Hydraulics Centre of the Danish Hydraulic Institute.

REFERENCES

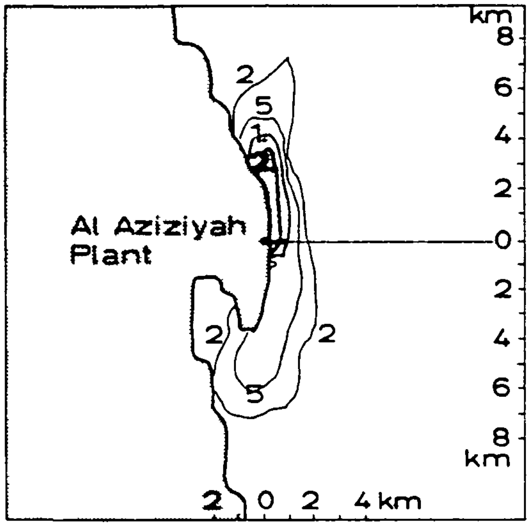
1. Abraham, G. 'Methodologies for Temperature Impact Assessment'. European Course on Heat Disposal from Power Generation in the Water Environment. Delft Hydraulics Laboratory, Delft, the Netherlands, 1975.

2. Dahl Madsen, K.I., "Modeling the Influence of Thermal Effluents on Ecosystem Behaviour". This Conference.

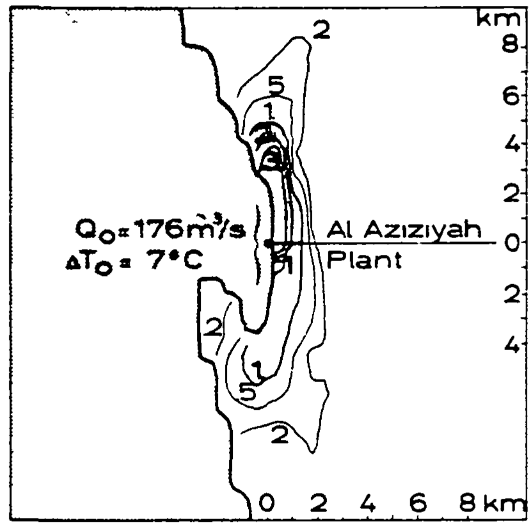
3. Abbott, M.B., "Computational Hydraulics: A Short Pathology". Journal of Hydraulic Research 14 (1976) No. 4.

4. Christensen, O. and Prahm, L.P., "A Pseudospectral Model for Dispersion of Atmospheric Pollutants". Danish Meteorological Institute, Lyngbyvej 100, Copenhagen, Denmark. 1976.

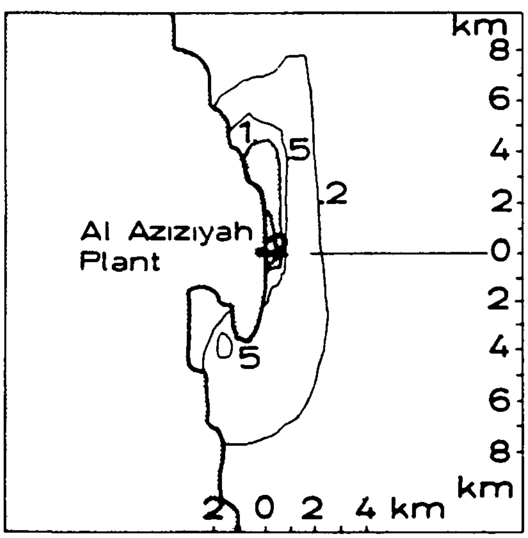
5. Schrøder, H., "Selection of Alternative Coastal Locations". This Conference.



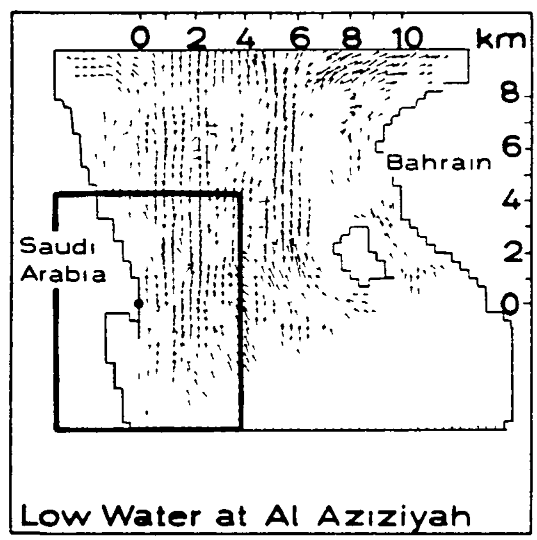
(a) most correct solution
- mass term included
- no numerical dispersion



(b) - mass term not included
- no numerical dispersion

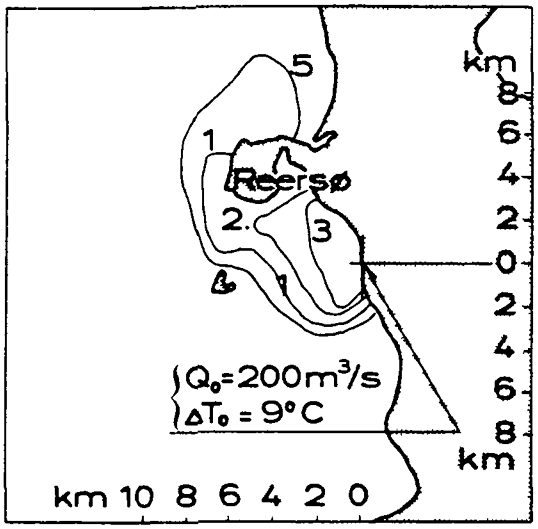


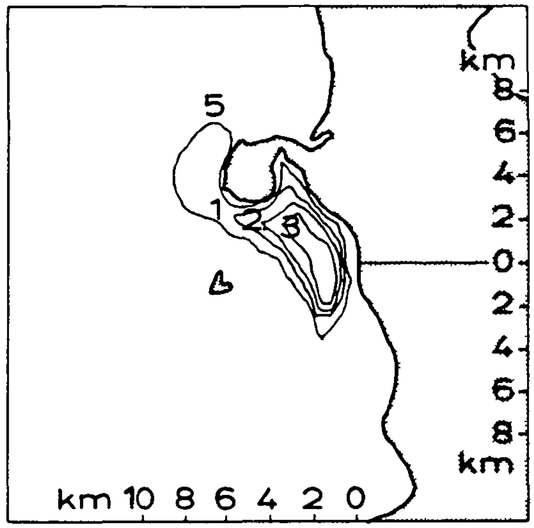
(c) - mass term not included
- with numerical dispersion



(d) - flow pattern in the strait between Saudi Arabia and Bahrain

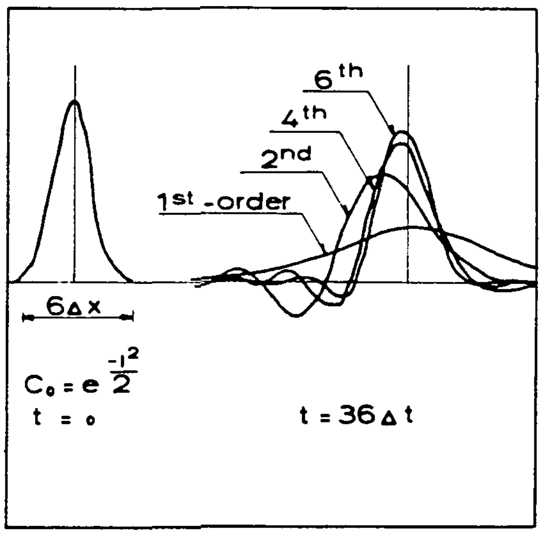
Fig. 1: Computed Temperature Contour Lines for the Al Aziziyah Desalination and Power Plant, with Different Types of Errors.





(a) Near-field: Eight meshes (b) Near-field: One mesh

Fig. 2: Influence of near-field distribution on far-field contours.



$C_0 = e^{\frac{-l^2}{2}}$

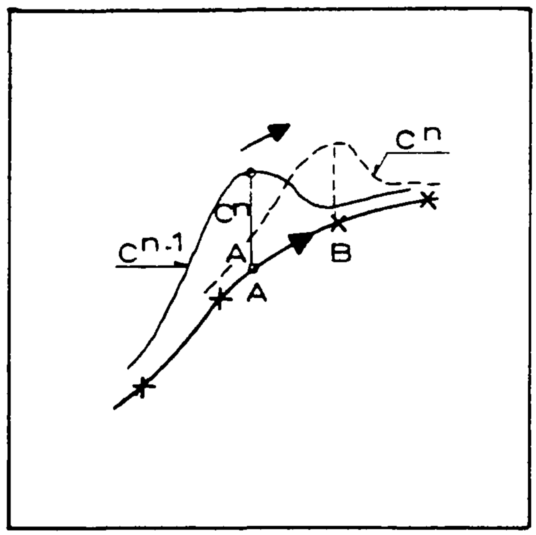


Fig. 3: Examples of Errors in Difference Approximations to Transport-Dispersion Equations.

Fig. 4: Principle of 12-Point Scheme.

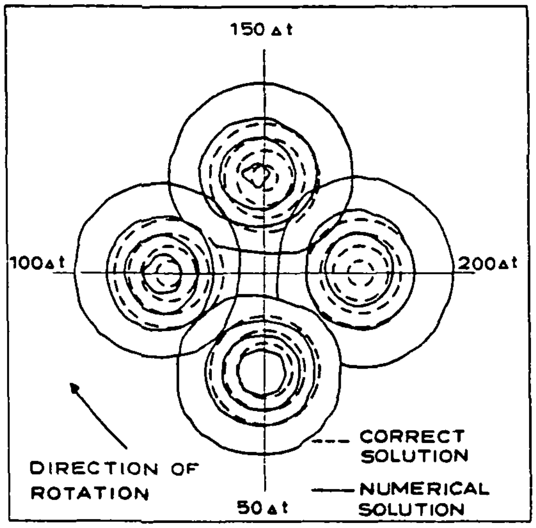
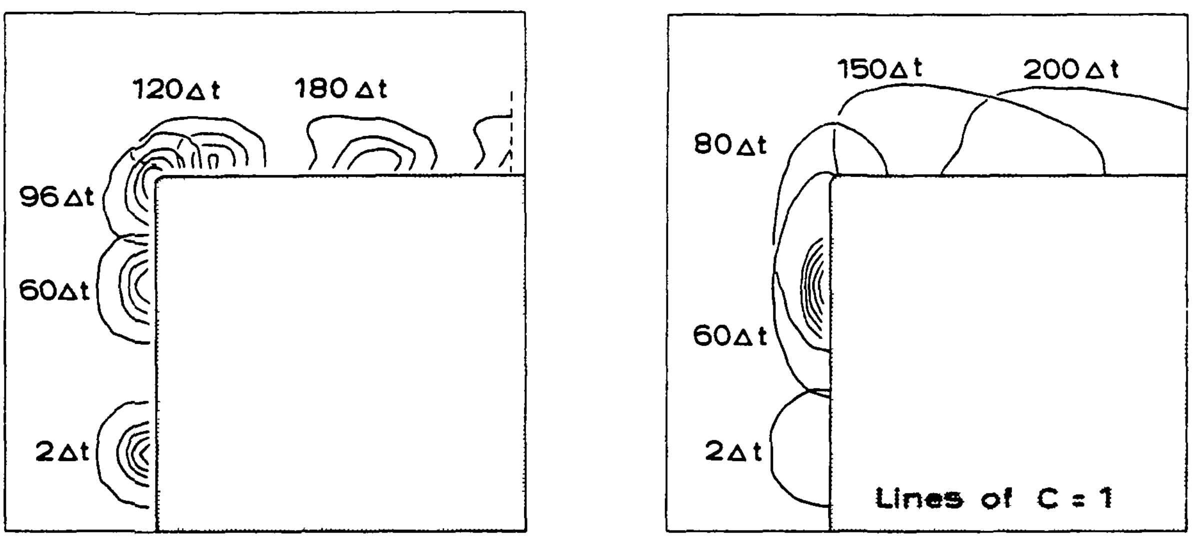


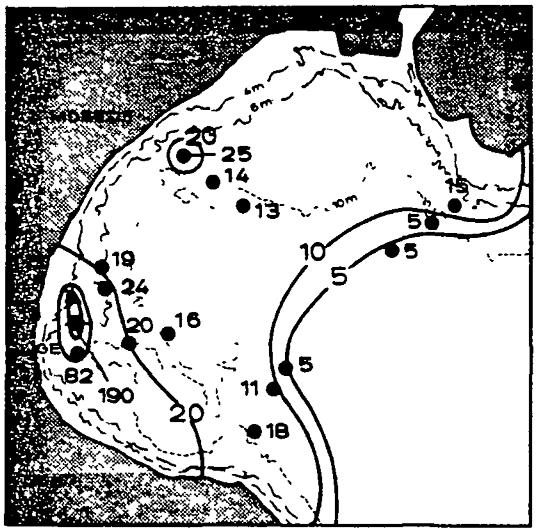
Fig. 5: Rotation Test,
12-Point Scheme.



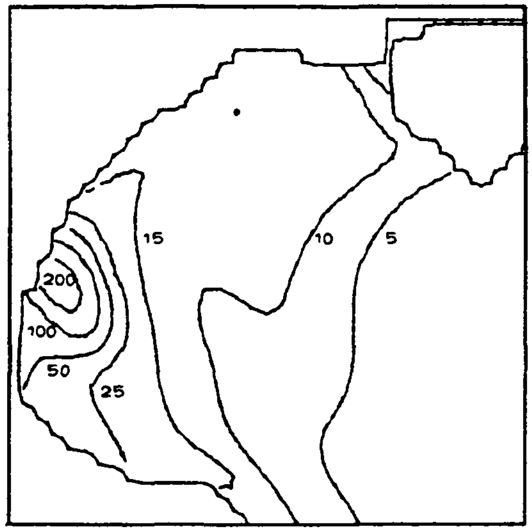
(a) 12-Point Scheme

(b) Scheme with Numerical dispersion

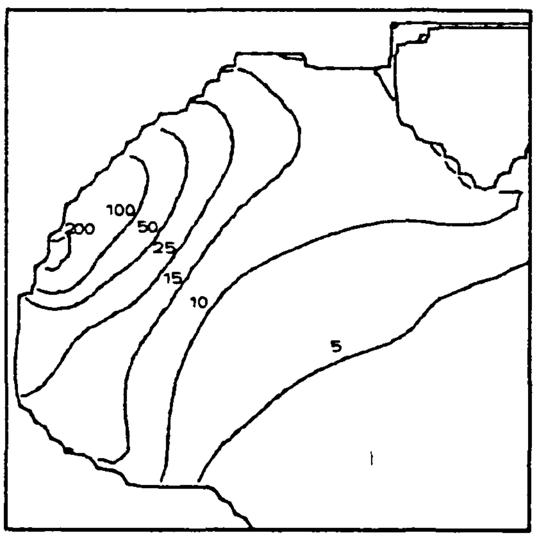
Fig. 6: L-Test of Different Difference Schemes.



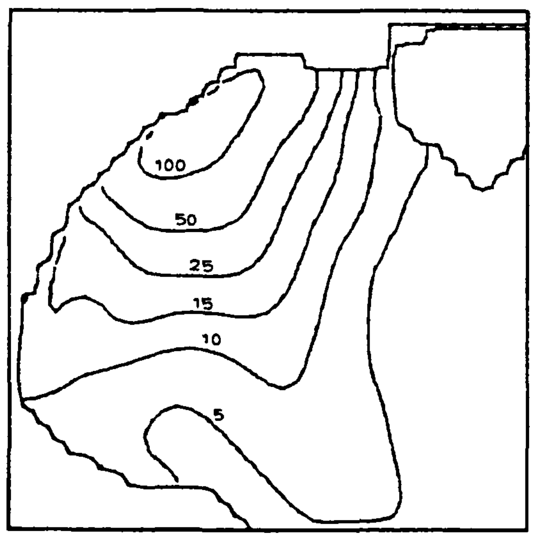
(a) - recorded 0 hrs



(b) - computed 23 hrs

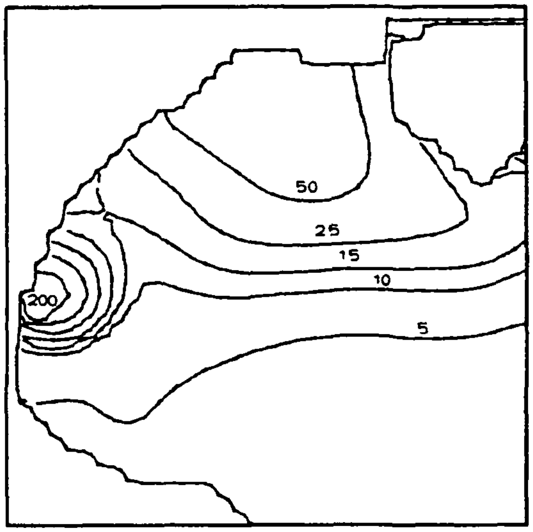


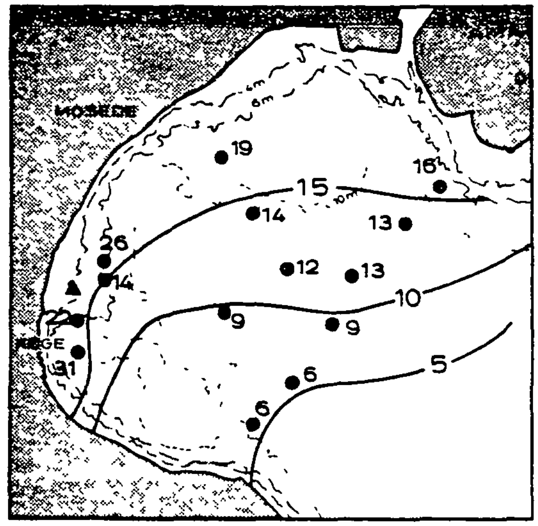
(c) - computed 71 hrs



(d) - computed 111 hrs

Fig. 7. a-d: Sequence of Recorded and Computed Concentration Contours for Køge Bay.
Source: 16 hrs on, 8 hrs off per 24 hrs, with 2 x 24 hrs off during weekend between 88 hrs and 144 hrs.
Load: 168 gram/sec.

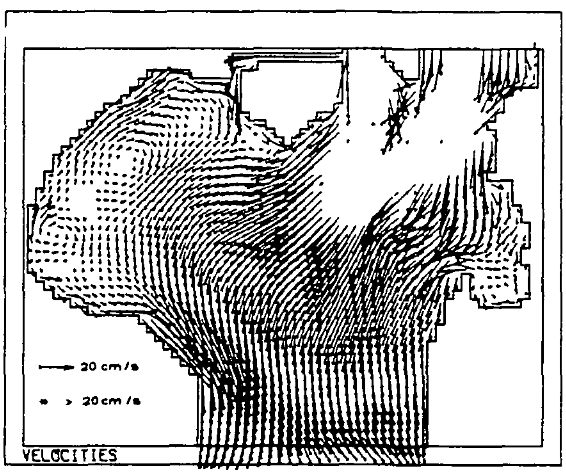


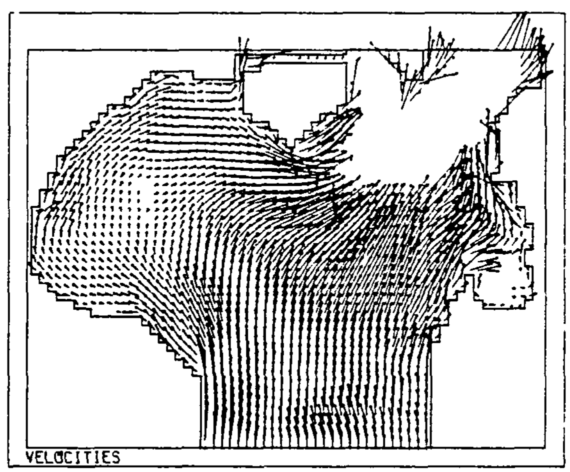


(e) - computed 168 hrs

(f) - recorded 168 hrs

Fig. 7. e-f: Sequence of Recorded and Computed Concentration Contours for Køge Bay, Continued.





(g) - typical flow pattern at 27 hrs, southgoing current in the Sound, westerly winds 10-15 m/s

(h) - typical flow pattern at 51 hrs, northgoing current in the Sound, southerly winds 8-10 m/s

Fig. 7. g-h: Typical Flow Patterns.