Stellingen


1. The Vertical-Horizontal Splitting technique can be applied for three-dimensional numerical modelling of shallow water flow. Models based on this technique can be effective, efficient and require limited computer resources.

2. The vertical distribution of velocities in shallow water flows is highly influenced by wind action. The vertical distribution of the eddy diffusivity in such flows can be approximated using a conventional parabolic law, with a friction velocity equal to the larger of the bed- and surface-friction velocities.

3. Two-dimensional depth-averaged numerical flow models are inappropriate for simulating wind-induced flows, as the bottom friction coefficient is a function of the wind and flow conditions. (Quasi-)three-dimensional modelling is preferable for such flows.

4. When applying a particle-tracking model of shear dispersion in shallow water, the maximum length of random displacement can be equated to the water depth. The resulting inaccuracy in the computed concentration distributions is then within acceptable limits.

5. It is meaningless to think of (computer) model validation as a kind of seal of approval for a given model that can be applied once and for all ... ... Validation must be viewed as a process that accompanies the development, implementation, and operational use of the model.


6. The last thing people are worried about, while starving, is the issue of environmental problems.

7. Commercialization of universities induces an increase in the number of scientific publications from these institutions, but also a decrease in the quality of these publications.

8. The complexity of the Chinese language is often wrongly attributed to the thousands of characters used. These characters are in fact words with specific meanings. Besides, less than ten basic elements - comparable with letters in the English alphabet - are found in the construction of these characters.

9. Progressive political and economic reform is probably the best way towards prosperity for an immense country like China, because of its enormous traditional and cultural inertia.

10. Professional tennis competitions are nowadays in danger of becoming contests of sheer physical power. Imposing an upper limit on the maximum speed of service may save this sport.
Quasi-Three-Dimensional Numerical Modelling of Flow and Dispersion in Shallow Water

Proefschrift

ter verkrijging van de graad van doctor aan
de Technische Universiteit Delft,
op gezag van de Rector Magnificus,
prof. drs. P.A. Schenck,
in het openbaar te verdedigen
ten overstaan van een commissie
aangewezen door het College van Dekanen
op 4 mei 1993 te 16.00 uur

door
Xian-You Jin,
geboren te Hubei, China,
civiel ingenieur.
Dit proefschrift is goedgekeurd door de promotoren:
prof. dr. ir. J.A. Batjes

Toegevoegd promotoren:
dr. ir. C. Kranenburg
To my wife Ping-Ping and my parents
CONTENTS

SAMENVATTING ........................................................... viii
SUMMARY ................................................................. ix
ACKNOWLEDGEMENTS ................................................... x
LIST OF MAJOR SYMBOLS ............................................... xi

CHAPTER 1 - INTRODUCTION .............................................. 1

CHAPTER 2 - EXISTING FLOW AND DISPERSION MODELS ................. 5
  2.1 The background of quasi-3D flow modelling ....................... 5
  2.2 Transport models of dissolved matter ............................. 10

CHAPTER 3 - A QUASI-3D NUMERICAL FLOW MODEL .................. 13
  3.1 Introduction ........................................................ 13
  3.2 Main features of DUTRID ......................................... 14
  3.3 The shallow water equations ...................................... 16
    3.3.1 Introduction .................................................. 16
    3.3.2 The hydrostatic pressure distribution ....................... 16
    3.3.3 The shallow water equations ................................ 17
  3.4 The depth-integrated flow ........................................ 21
    3.4.1 The depth-integrated flow module .......................... 21
    3.4.2 The effect of advective terms .............................. 23
  3.5 The velocity profile module ...................................... 25
    3.5.1 Basic equations .............................................. 25
    3.5.2 Boundary conditions .......................................... 27
    3.5.3 Finite-difference equations ................................. 31
    3.5.4 Numerical approximation of the advective terms .......... 34
  3.6 Computation procedure ............................................ 36
    3.6.1 Coupling of modules ....................................... 36
3.6.2 Computation of the bottom shear stress ........................................... 38
3.6.3 Correction of velocity profiles .......................................................... 39
3.7 Summary ................................................................................................. 40

CHAPTER 4 - A STUDY OF THE VERTICAL EDDY VISCOSITY .................. 43
4.1 Introduction ........................................................................................... 43
4.2 The experiment of Yu (1987) ................................................................. 44
4.3 Computations using DUTRID ................................................................. 45
4.4 Computations using the k-ε model ........................................................ 51
4.5 Conclusions and discussion ................................................................... 55

CHAPTER 5 - TEST COMPUTATIONS WITH DUTRID ............................... 57
5.1 Introduction ........................................................................................... 57
5.2 Uni-directional flows ............................................................................ 58
  5.2.1 Pressure-gradient driven flow ......................................................... 58
  5.2.2 Surface shear induced flow ............................................................ 60
  5.2.3 Wind-driven flow in a closed channel ............................................ 62
  5.2.4 Flow over a bed of nonuniform slope ........................................... 64
5.3 Secondary current in nearly horizontal flow ......................................... 67
5.4 Flow in a model tidal harbour basin ..................................................... 70
5.5 Conclusions ........................................................................................... 78

CHAPTER 6 - A PARTICLE MODEL FOR DISPERSION OF SOLUTES ........ 81
6.1 Introduction ........................................................................................... 81
6.2 The particle tracking model .................................................................. 83
  6.2.1 Theoretical formulation ................................................................. 83
  6.2.2 General structure of the model ...................................................... 88
6.3 Modelling turbulent diffusion ............................................................... 89
  6.3.1 Homogenous turbulence ............................................................... 90
  6.3.2 Nonhomogenous turbulence ......................................................... 92
6.4 Modelling advection ............................................................................. 94
  6.4.1 Cell-analytical solution method ..................................................... 94
  6.4.2 The Molenkamp test .................................................................... 97
  6.4.3 Advection in vertical planes .......................................................... 100
6.5 Two-dimensional shear dispersion ......................................................... 101
  6.5.1 Theory .......................................................................................... 102
6.5.2 A numerical example ........................................ 104
6.5.3 Shear dispersion in shallow water flow .................... 106
6.6 Summary and conclusions ...................................... 109

CHAPTER 7 - MODELLING DISPERSION OF A SOLUTE IN LAKE YSSEL 111
7.1 Introduction ..................................................... 111
7.2 Field experiment in Lake Yssel ................................. 113
  7.2.1 General description of the experiment ...................... 113
  7.2.2 Hydrodynamic conditions during the experiment ........... 115
  7.2.3 The observations .......................................... 119
7.3 Simulation of flow in Lake Yssel ............................... 121
  7.3.1 Computational grid ....................................... 122
  7.3.2 Model parameters ........................................ 122
  7.3.3 Circulations induced by constant wind .................... 125
  7.3.4 Simulation of flow during the experiment ................. 134
7.4 Simulation of dispersion of solute ............................ 135
  7.4.1 Particles representation of amount of solute .............. 136
  7.4.2 Simulation of dispersion of solute ....................... 136
  7.4.3 Sensitivity study ........................................ 147
  7.4.4 Comparison with 2D computations ........................ 154
7.5 Discussion and conclusions ................................... 157

CHAPTER 8 - SUMMARY AND CONCLUSIONS .......................... 161

REFERENCES ......................................................... 167

CURRICULUM VITAE .................................................. 175
Samenvatting

Een quasi-3D numeriek model voor grootschalige stromingen opgewekt door wind en getij in ondiepe wateren zoals meren, estuaria en kustgebieden is ontwikkeld. Eventuele dichtheidsverschillen zijn verwaarloosd en een hydrostatische drukverdeling wordt verondersteld.

In het model worden de stroomsnelheidsverdelingen in het horizontale vlak en in de verticale richting berekend in twee verschillende modules. De twee modules zijn gekoppeld via het verhang van het vrije oppervlak en de bodemschuijspansing. Aan de bodem wordt de kleefconditie opgelegd; de bodemschuijspansing wordt berekend met behulp van de wandwet. De advectieve termen worden in beide modules in rekening gebracht.

Het model wordt toegepast op verschillende stromingen, o.a. op stromingen in een vierkante haven aan een getijderivier. De numerieke resultaten komen goed overeen met bestaande metingen.

De grootte van de verticale turbulente viscositeit in door wind aangedreven stromingen is onderzocht. De conventionele parabolische verdeling over de verticaal, met een schuijspansingssnelheid gelijk aan de grootste van die aan bodem en vrij oppervlak, geeft goede overeenstemming tussen numerieke resultaten en de metingen van Yu (1987). De resultaten zijn geverifieerd met een $k-\varepsilon$ model.

Een deeltjesmodel is ontwikkeld voor de verspreiding van passieve opgeloste stoffen in stromingen. Het model is gebaseerd op de randomwalk theorie. In het model wordt de advectie per cell analytisch opgelost. Turbulente diffusie is gemodelleerd door de deeltjes random verplaatsingen op te leggen.


Het stromingsmodel en het deeltjesmodel worden toegepast voor de simulatie van de verspreiding van een opgeloste stof in het IJsselmeer. Windschuijspansingen zijn berekend uit waarnemingen van windsnelheden. De horizontale uitwisselingscoëfficiënt en de bodemruwheid zijn constant verondersteld. Uit de numerieke resultaten blijkt dat ercomplex horizontale stromingspatronen in het meer optreden. De variaties van de stroomsnelheden in de verticale richting zijn aanzienlijk. Goede overeenstemming is gevonden tussen numerieke resultaten en de metingen aan de concentratieverdelingen in het meer.

Een gevoeligheidsonderzoek toont aan dat de concentratieverdeling sterk afhangt van de wind en de horizontale uitwisselingscoëfficiënt. De invloed van de sub-grid-scale turbulentie blijkt groot te zijn. De gebruikte roosterafstand laat slechts een beperkte weergave toe van topografische details, zoals die van het Ketelmeer.
Summary

An efficient quasi-3D numerical model of large-scale circulations induced by wind and tidal forces in shallow waters such as lakes, estuaries and coastal zones, has been developed. The water system considered is homogeneous and the pressure distribution is assumed to be hydrostatic.

The velocity distributions in the horizontal plane and in the vertical direction are computed separately in two modules (Vertical-Horizontal Splitting). These modules are coupled via the free surface gradients and the bottom shear stresses. A no-slip condition is assumed at the bottom, and the bottom shear stresses are computed using the law of the wall. The advective terms are retained in both modules.

Applications of the model to a number of flow cases are presented, among which 3D circulations in a square harbour basin opening to a tidal river. The numerical results compare well with available experimental evidence.

A study of the vertical eddy viscosity in the wind-driven flow in a laboratory flume is presented. The conventional parabolic distribution over the depth is found to give a remarkably good fit of velocity distributions to measurements of Yu (1987), provided the friction velocity is equated to the larger one of the bed- and surface-friction velocities. This result is supported by computations using a k-ε model.

Based on the random walk theory, a particle tracking model simulating dispersion of a passive solute in water flow has been devised. Advection is simulated using a cell-analytical approach. Turbulent diffusion is simulated through random displacements of the particles.

The particle model is applied to 2D shear dispersion in shallow water flow. The results compare well with analytical solutions given by Taylor (1954) and Elder (1959). The study shows that the maximum length of vertical random displacements can be equated to the local water depth rather than the vertical grid spacing. The resulting error in predicted concentrations is within acceptable limits for engineering applications.

The flow and dispersion models are applied to simulate a field dye dispersal experiment in Lake Yssel in the Netherlands. Wind shear stresses are computed from wind records. Constant values are assumed for the horizontal eddy viscosity and for the bottom roughness height. Fair agreement is obtained between numerical and experimental results for the concentration distributions.

Numerical results for the flow in the lake show complex variations of the water velocities in the horizontal plane and marked variations over the water depth. Sensitivity studies show that the results for the concentration distributions markedly depend on the horizontal eddy viscosity and the wind condition. The influence of the sub-grid-scale turbulence seems to be large. The grid size used results in an insufficient representation of topographical details, such as those of the connected Ketelmeer.
Acknowledgements

This work has been completed during my employment as a research assistant in the Hydromechanics Section of the Hydraulic and Geotechnical Engineering Division, Faculty of Civil Engineering, Delft University of Technology.

First of all, I wish to thank Dr C. Kranenburg who has provided continuous guidance throughout this study. His vast experience in fluid dynamics and his readiness to share that experience with his colleagues have been an ever-reliable resource when I was faced with difficulties in the research. I am grateful for his abundant advises, valuable suggestions and continuous encouragement.

I would like to thank Prof Dr J.A. Battjes who has been the supervisor of this study. His sharp physical insight into problems related to fluid mechanics and his exemplary critical attitude towards scientific research in general, have always been a source of inspiration, and undoubtedly will continue to be so in my future career working in the fascinating, sometimes also intriguing, field of fluid mechanics.

Thanks are extended to Prof Dr J.P.Th. Kalkwijk who has given me this chance to make the study in this group, and for his guidance during the initial phase of this study.

I am indebted to Dr N. Booij for helping me with the numerics. Prof Dr G.S. Stelling is acknowledged for making available the theory of the cell-analytical method used for particle tracking, and for many discussions.

I wish to express my appreciations to the direct or indirect support for this work from all the colleagues in the Hydromechanics Section. Special thanks are due to young researchers in the Hydraulic and Geotechnical Engineering Division, with whom I came into daily contact: Arris Tijsseling, Pieter Blom, Ed Langendoen, Arno Talmon, Gijs Hoffmans, Daan Dunsbergen and Peter de Wit. Their enthusiasm and their friendship have made my stay in this group a pleasant experience.

This work was partly financed by Rijkswaterstaat. Rijkswaterstaat-RIZA and N.V. Watertransportmaatschappij Rijn-Kennemerland are acknowledged for making available the data of the dye dispersal experiment in Lake Yssel.
List of Major Symbols

\[ \begin{align*}
  c & = \text{concentration;} \\
  C & = \text{coefficient of Chézy;} \\
  c_d & = \text{drag coefficient;} \\
  C_o & = \text{Coriolis parameter;} \\
  D & = \text{water depth;} \\
  D_y & = \text{diffusivity tensor;} \\
  D_{xx}, D_{xy}, \ldots & = \text{differential convection terms;} \\
  E & = \text{horizontal eddy viscosity;} \\
  E_{\text{ed}} & = \text{viscosity related to three-dimensional turbulence;} \\
  E_{\text{SGS}} & = \text{viscosity related to sub-grid-scale turbulence;} \\
  f, f_r, F & = \text{functions;} \\
  F_r & = \text{friction coefficient;} \\
  Fr & = \text{Froude number;} \\
  g & = \text{acceleration due to gravity;} \\
  H_i & = \text{level of the free water surface;} \\
  H_b & = \text{level of the bottom surface;} \\
  k & = \text{height of bed roughness element;} \\
  K & = \text{dispersion coefficient;} \\
  L & = \text{characteristic length-scale;} \\
  P & = \text{air pressure;} \\
  p & = \text{hydraulic pressure;} \\
  Q & = \text{flow rate per unit width (vector);} \\
  Ro & = \text{Rossby number;} \\
  T & = \text{bottom shear stress vector;} \\
  t & = \text{time;} \\
  U_a & = \text{wind speed;} \\
  U, V, W & = \text{characteristic values of flow velocity components;} \\
  \mathbf{u}, \mathbf{v}, \mathbf{w} & = \text{velocity components in x-, y- and z-directions;} \\
  \bar{u}, \bar{v} & = \text{depth-averaged velocities in x- and y-directions;} \\
  u', v', w' & = \text{turbulent components of velocities;} \\
  u^* & = \text{friction velocity;} \\
  u_0^* & = \text{reference friction velocity;} 
\end{align*} \]
\[ u_a^* = \text{surface-friction velocity.} \quad W = \rho_a u_a^* |u_a^*| ; \]
\[ u_s^* = \text{surface-friction velocity.} \quad W = \rho u_s^* |u_s^*| ; \]
\[ u_b^* = \text{bed-friction velocity}; \]
\[ W = \text{surface wind shear stress vector}; \]
\[ x, y, z = \text{Cartesian coordinates}; \]
\[ z_0 = \text{equivalent bottom roughness height}; \]
\[ \Delta t = \text{time-step}; \]
\[ \Delta x = \text{horizontal grid-spacing}; \]
\[ \tau_x, \tau_y = \text{turbulent shear stress components}; \]
\[ \rho_a = \text{air density}; \]
\[ \rho = \text{water density}; \]
\[ \Omega = \text{angular speed of the earth}; \]
\[ \varphi = \text{latitude}; \]
\[ \kappa = \text{Von Karman coefficient (≈ 0.4)}; \]
\[ \nu = \text{vertical eddy viscosity}. \]

**Subscripts**

\[ x, y = \text{components in the corresponding directions}; \]
\[ b = \text{value at the bottom}; \]
\[ s = \text{value at the free surface}. \]

**Superscript**

\[ - = \text{depth-averaged value}. \]
CHAPTER 1

INTRODUCTION

Water is one of the basic natural resources of human beings. The conservation and protection of our water environment is therefore of paramount importance to our well-fare or even to our very existence. Since the last century, however, environmental pollution caused by human activities has been accumulating. Nowadays, the natural environment has so much degenerated that environmental protection has become a subject of major social and economic concern world-wide.

Industrialization has brought great social progress, but it also produces huge quantities of waste materials that have to be disposed of. Other human activities also generate pollutants such as sewage water. Fine sediments may also become a hazard to the water quality because they absorb micro-pollutants. Because of their sheer quantity, most of these materials can only find their place in the water environment and possibly be digested thereby through chemical and biological actions, which is one of the reasons why it is at present still believed that "the best solution to pollution is dilution" (Fischer et al, 1979). While ways of reducing waste discharges are being searched and put into application, research into the physical mechanism of transport of pollutants in natural waters has also intensified during the last few decades. Considerable understanding has been gained through physical and computer modelling as well as through field experiments, leading to better control of waste disposal.

In general, the transport of pollutants in natural waters is a compound process involving physical, chemical and biological actions. Physically, pollutants are transported and dispersed essentially by water flows mostly induced by wind or/and tidal forces. Wind waves do not directly contribute to transport of materials, although they generate turbulence in water and interact with water flows, thereby indirectly become involved in the entire dispersion process.
Water flows can be generally classified into two major categories, namely the small scale flow and the large scale water flow. As examples of flows in the first category we may mention flows over trenches in the sea bottom, flows around fixed or floating structures, flows near intake or discharge points of a lake or a reservoir and flows in a river bend. The common characteristic of such flows is that the length scales of the flow field in the horizontal and vertical directions are of the same magnitude. As examples of flow in the second category we may mention flows in the so-called shallow waters such as lakes, reservoirs, estuaries and coastal zones where the horizontal length scale is much larger than the vertical one. The large scale flow to a large extent determines the fate of discharged materials.

With the advent of computers, prediction of water flow and dispersion of waste materials through the flow using numerical models has become possible. Along with the rapid development of modern computer technology, computers become nowadays faster and cheaper, and capable of storing and processing large quantities of data. Under the impetus of this development, various two and three-dimensional numerical models have been developed that simulate and predict water flow and dispersion of materials in natural water bodies. One finds a great number of two-dimensional models in the literature. Being used as a tool for engineering design and as part of decision support systems, some of these models have found their way into engineering applications as well. Three-dimensional modelling, on the other hand, is still at its developing stage.

Because a water flow model is often used in combination with transport models to study, for example, the dispersion of suspended or dissolved matters, detailed information on the three-dimensional distribution of flow quantities is needed. Three-dimensional numerical flow models are then indispensable. On the other hand, pollutant and sediment transport processes often involve periods of months or even years. Fully three-dimensional flow modelling for such long periods on modern computers is still expensive in CPU time and therefore not always feasible. Consequently, while fully three-dimensional models are being developed, quasi-three-dimensional numerical models are often found useful because they are cheaper and more efficient.

The basic feature of quasi-three-dimensional modelling is perhaps the application of the Vertical-Horizontal Splitting (VHS) algorithm. By using this algorithm the dependencies of
flow quantities on the horizontal and vertical coordinates are determined separately. As a result, the system of finite difference equations to be resolved is relatively small so that higher computational efficiency is obtained. It can be shown that the separation (or splitting) is possible for flows in shallow waters where a hydrostatic pressure distribution can be assumed.

The aim of the present study is to devise computer models which can effectively and efficiently be applied for modelling three-dimensional flow and dispersion in shallow water.

Water flows induced by wind and tidal forces are considered. Wind waves are not modelled, but their contribution to the flow through turbulence generation can be taken into account. Density stratification is not considered either, although horizontal density gradients can be incorporated. The transport of substances through shallow water flows is also investigated. The substances are assumed dissolved and passive.

An efficient quasi-three-dimensional model for large scale flows in shallow waters has been developed. It is composed of two coupled modules. In one module the depth-averaged shallow-water equations are solved; in the second module the vertical distribution of horizontal velocities is determined. The two modules are coupled by the free surface gradients and the bottom shear stresses.

To model dispersion of dissolved matter in shallow waters, a particle tracking module running in conjunction with the quasi-three-dimensional flow model, has also been developed. A complete dispersion model for passive pollutants is thereby devised. As a practical application, the dispersion model has been applied to a field dye dispersal experiment carried out in Lake Yssel in the Netherlands. The experiment covered a period of approximately two months and a large part of the lake surface.

The organization of this dissertation is as follows. Some general aspects of existing flow and dispersion models are discussed in chapter 2. In chapter 3 the structure as well as the main characteristics of the flow model developed in this study are outlined, followed by a detailed description of the model. A study of the vertical eddy viscosity using laboratory measurements and numerical models are presented in chapter 4. Some computational examples with which the flow model was tested are given in chapter 5. The particle tracking
model is presented in chapter 6, while chapter 7 deals with the numerical simulations of the dye dispersal experiment carried out in Lake Yssel. Summary and conclusions from the present study are presented in chapter 8.
CHAPTER 2

EXISTING FLOW AND DISPERSION MODELS

2.1 The background of quasi-3D flow modelling

Shallow water flows are found in a wide variety of natural water bodies such as shallow lakes, reservoirs and the coastal zone. The common characteristic of such water bodies is that the water depth is much smaller than a typical length in the horizontal directions. The flows in a water body are driven in most cases by wind shear stresses exerted on the free water surface, tide-induced water level variations at the boundaries, and the atmospheric pressure gradient. Density stratification, river inflows and outflows also contribute to the large-scale flow.

Numerical modelling of free-surface water flows began around the sixties, and research in this area has intensified during the past two decades. Despite its short history, one finds at present an overwhelming quantity of literature concerning water flow modelling. One may refer to several review and 'state-of-the-art' publications: Mortimer (1974), Csanady (1975), Bengtsson (1978), Simons (1980), Lindijer (1981) and more recently Nihoul and Jamart (1987).

Numerical models of shallow water flows range from one-dimensional models (1D), two-dimensional vertical (2DV) models, two-dimensional horizontal (2DH) models to three-dimensional (3D) models. In a 1D model, flows are considered as channel flows that are uniform in the cross-section, or a network of such flows. A 2DH model predicts variations of depth-averaged flow velocities in the horizontal plane. The vertical distribution of flow velocities can be determined by using a 2DV model, although only laterally uniform flow can be considered herewith. 1D, 2DH and 2DV models are relatively simple, cheap and can
provide useful results for engineering applications. Nevertheless, their applications are limited to situations where the contribution of the 3D characteristics of flow has no significance to the problem at hand.

In flows found in practice, however, marked 3D flow characteristics have often been observed. It is well known that in wind-induced flows in lakes, both the magnitude and the orientation of the flow velocity vector change markedly in the vertical direction. A 2DH or 2DV model can only yield rough approximations to such flows. On the other hand, detailed knowledge of the velocity profile in the vertical is of key importance for solving a large variety of practical problems. For example, the transport of silt and other fine sediments along the bottom is determined by the local flow velocities near the bottom. The spreading of light contaminants such as oil on the free water surface depends on the flow velocities at the free water surface. The complete vertical velocity profiles can play an important part when predicting the dispersion process of dissolved substances such as dissolved contaminants in water, or the resuspension and transport process of solid particles. Obviously, 3D flow models are preferred in such cases.

One can find at present a large number of 3D models in the literature. These models can roughly be classified into three major categories, namely, fully analytical models, fully numerical models and quasi-3D (Q3D) models.

Fully analytical models

In case of very simple and idealized horizontal geometry and bottom topography, one can find analytical solutions to the basic flow equations provided that certain hypotheses are made concerning the flow characteristics, such as the turbulent shear stress in the flow and at the boundaries. This approach has often been used to analyze the effect of density stratification on flows. This effect is assumed to dominate over effects of other aspects of the flows (Csanady, 1972; Jacobs, 1974). For example, turbulent exchanges in the vertical direction as well as frictions at the boundaries are usually neglected. This type of models can be used as testing cases for numerical models. However, it generally has little significance for practical applications.
Fully 3D numerical models

In this category of flow models the dependence of flow velocities on time and the three spatial coordinates is resolved numerically. Following the methods used for the solution in the vertical direction, two different types of 3D models can further be distinguished: 'Multi-layer Models' and 'Multi-level Models'.

In a Multi-layer Model (Simons, 1973, 1980; Vreugdenhil, 1978, 1979), the water body is divided into a number of homogeneous layers on top of each other, each layer having different flow characteristics such as density and turbulence intensity. The interfaces between layers are movable. The velocities and density within each layer are averaged in the vertical direction. The set of 2D flow equations for each individual layer, together with the kinematic and hydrodynamic boundary conditions at the interfaces between layers, determine the complete flow field. Multi-layer models are typically applied to simulate stratified flows in deeper water.

In a typical Multi-level model, the mathematical-numerical treatment of the 3D flow equations in all three directions is identical (Ligget, 1970; Simons, 1976, 1978; Leendertse and Liu, 1975, 1977; Uittenbogaard et al, 1991). The hydrostatic pressure distribution may also be assumed in such models (Uittenbogaard et al, 1991).

With the rapid development of modern computer technology, computers become nowadays faster and cheaper, and capable of storing and processing large quantities of data. 3D flow modelling is no longer unrealistic on modern computers.

However, practical applications of fully 3D flow models is still inefficient and expensive in CPU time, especially because these applications - for example, the prediction of long-term morphological evolution of the coastal zone caused by sediment transport - often involve periods of months or even years. For a typical rectangular flow field of 400 km × 800 km × 65 m discretized into a lattice of 10 × 20 × 10 grid points, a typical 3D model needs 1 hour 47 minutes CPU time for the simulation of a period of 30 days or 22 hours of CPU times for one year on an Amdahl 5850. For equivalent computations, a typical two-dimensional model would need only one tenth of the CPU time (Lardner and Cekirge 1988).

One therefore still needs to look for cheaper and more efficient 3D numerical models.
that nevertheless provide sufficient information on the water flows. This can be achieved by making approximations in the model equations so that efficient numerical algorithms can be applied. For large-scale flows as considered in the present study, the shallow water conditions are satisfied, which makes Q3D flow modelling possible.

Quasi-3D numerical models

In Q3D numerical models the basic flow equations are simplified by using the shallow water assumptions. The vertical acceleration is assumed much smaller than the acceleration due to gravity so that a hydrostatic pressure distribution is satisfied. Although the definitions of Q3D numerical flow models found in the literature are often ambiguous, there are some common features in such models, namely, the assumption of a hydrostatic pressure distribution and the application of the Vertical-Horizontal Splitting algorithm (VHS). By using the VHS algorithm the solution of the flow quantities is made in two separate steps, one dealing with distributions of the state variables in the horizontal plane and the other with distributions in the vertical direction.

Depending upon the way the vertical structure of the flow field is determined, two types of Q3D flow models can further be distinguished, namely: Q3D semi-analytical models and Q3D numerical models.

Some Q3D flow models are labelled 'semi-analytical' in the sense that in these models, velocity profiles in the vertical direction are obtained analytically, while the dependencies of the state variables upon time and the horizontal coordinates are determined numerically by using grid discretization.

In most Q3D semi-analytical models the nonlinear horizontal advection terms and diffusion terms in the basic equations are neglected. Also, the boundary conditions are often simplified, for example, by using a linear formulation of the bottom shear stresses. By further assuming simple vertical eddy viscosity distributions, analytical solutions of the vertical profiles of horizontal flow velocities can directly be obtained (see, for example, Van der Giessen et al (1988) in which a constant eddy viscosity is assumed. A linearized bottom
stressed formulation is applied in the model as well).

The so-called modal or spectral models often reported in the literature can also be classified as semi-analytical Q3D models (Cheng, 1977; Davies, 1980, 1987; Zitman, 1992). In these models the flow velocities are expressed formally as a linear combination of a series of orthogonal functions ("shape functions"). These functions are usually taken from the eigenfunctions of the vertical diffusion operator. The formal solution with a finite number of shape functions is substituted into the basic equations, then integrated over the vertical direction. The resulting residual functions are then minimized using the Galerkin technique. A set of independent equations ("modal equations") thereby results, from which the unknown weighting coefficients in the formal solution can be determined. The eigenfunctions need to be determined only once for every application, either analytically (for simple idealized vertical distributions of the vertical eddy viscosity) or numerically (for an eddy viscosity found in practice). The modal equations are solved in the horizontal plane by using finite-difference discretization or using a convolution technique (Davies, 1987).

Arbitrary distributions of the vertical eddy viscosity can be assumed in spectral models. However, the nonlinear advective terms and the horizontal diffusive terms are usually neglected. The inclusion of the advective terms entails large complexity in the mathematical formulation of the model and can cause a large increase in the computational cost (Davies, 1980). There are also uncertainties concerning the physically correct bottom and free-surface boundary conditions to be implemented in these models (Zitman, 1990).

In the other type of Q3D flow models, namely, Q3D numerical models, the vertical profiles of velocities are solved numerically by using a grid discretization, as for the solution of the basic equations in the horizontal plane. It is relatively easy to retain nonlinear horizontal transport terms in the basic equation for the vertical flow structure and especially, to use better approximations of the boundary conditions. A typical example of Q3D numerical model is the model developed by Lardner and Cekirge (1988). A modified version of this model was recently reported by Al-Rabeh and Gunay (1992a). In this model the shallow water equations are used to solve the depth-averaged flow velocities for each time-step of computation, after which the vertical profiles of velocities are determined separately at each grid point in the horizontal plane. Horizontal diffusion terms and the nonlinear
advective terms are neglected in the model equations. Linear and quadratic formulations of the bottom shear stresses are used as the bottom boundary condition. The model was applied to simulate circulation induced by seasonal wind in the Safaniya sea area of the Persian Gulf (Al-Rabeh and Gunay, 1992a).

The system of equations to be solved in a typical Q3D numerical flow model is considerably smaller than in a fully 3D model due to the application of the VHS algorithm. As a result, higher computational efficiency is obtained. Q3D flow models are nowadays still competitive compared with fully 3D models, because simulation of practical flows using the latter is still inefficient and expensive in computer time, as was seen above. The largest drawback of Q3D models is perhaps their limitation to nearly horizontal flows because of the hydrostatic pressure distribution assumed. However, such models can still be applied to flows in a wide variety of water bodies such as lakes, reservoirs and the coastal zone, where the hydrostatic pressure assumption is approximately satisfied.

2.2 Transport models of dissolved matter

The dispersion of dissolved matter such as pollutants or nutrients in water flows is a complex process. These substances are transported by the flow through advection and turbulence. Active substances may accumulate, decay and interact with each other during the transport process. They could even act upon the conveying flows by changing the flow conditions. For example, thermal discharges result in changes in water density and hence changes in the fluid flow; erosion and sedimentation could result in modification of the bottom topography of the flow field.

The advective transport of passive dissolved matter depends exclusively upon the mean flow velocities. The prediction accuracy of a dispersion model is therefore directly related to the resolution of the flow velocities. Obviously, 3D flow models are preferable in this respect.

The dispersion of substances in water flows can be described by the well-known
advection-diffusion equation. This equation can be solved by using FDM or FEM in a Eulerian grid, which is the classical approach used to model dispersion (Noye, 1987). One finds many models based on this approach in the literature (Leendertse, 1970; Nokes et al, 1984; Van Stijn and Praagman, 1988). Other types of dispersion models are based on the Lagrangian approach, or the particle tracking method (see, for example, Al-Rabeh and Gunay (1992b)). In such models the total mass of pollutant is represented by a large number of discrete point masses, or particles that represent equal amounts of matter. By tracking these particles in the fluid flow, the advection of the substances is determined. The contribution to the transport through turbulence can be modelled by adding random components to the advective displacements of the particles. It can be shown that the probability density function of the position of the particles satisfies the Fokker-Planck equation (Feller, 1971) which is analogous to the advection-diffusion equation. Through this analogy relations can be determined between the displacements of particles and the dispersal characteristics of the flow.

Compared with the Eulerian approach the particle tracking method has a number of interesting advantages:

* **no negative concentrations.**

With the Eulerian approach negative concentrations are often obtained in regions with large concentration gradients. As a result, a filter is needed to smooth the results. Negative concentrations never appear with the particle tracking method. This property is perhaps one of the most interesting features of a particle model. It makes the model particularly suited for modelling the initial mixing of discharged substances where large concentration gradients are usually found;

* **no numerical diffusion.**

In the Eulerian approach, numerical diffusion may be introduced through the numerical approximations of the advection terms in the equations of motion. This is avoided in a particle tracking model where no finite-difference equations are used. Therefore, the amount of diffusion in the numerical model is under exact control;

* **conservation of mass guaranteed.**

In the particle tracking method one can easily assure that particles representing
substance mass do not mistakenly disappear or be created. Therefore, the conservation of mass is guaranteed;

* easy to implement.

The particle tracking method is physically straight-forward. The mathematical-numerical formulation is considerably simpler than with a model that solves the advection-diffusion equation. Nevertheless, since the positions of particles in the numerical grid should be checked against grid boundaries and the boundaries of the flow field at each step of computation, the programming of such a model could be less easy.

Despite these interesting features, some disadvantages are inherent in the particle tracking method. The accuracy of predicted concentrations is obviously related to the number of particles used. A large number of particles may be needed to adequately represent the total mass of substances dispersed in a large flow field. In that case, a large storage capacity of computer is required, and the computation cost could be high. In practice, one should try different number of particles and find for each individual case a compromise between accuracy of computation and cost.

A particle tracking model can be used in combination with a conventional dispersion model based on the Eulerian approach. It can be used for modelling near-field mixing of a point discharge for which a conventional model is less appropriate. As long as the particle or solute cloud is small, the particle model can be used efficiently since the required number of particles is small. After the cloud is stretched over a large water surface (far-field dispersion), one may proceed with the simulation using the Eulerian model, with the final concentration distribution resulting from the particle model being used as the initial condition.
CHAPTER 3

A QUASI-3D NUMERICAL FLOW MODEL

3.1 Introduction

A new quasi-3D numerical model of shallow water flow called DUTRID was developed in the present study. The model consists of two coupled modules. Emphasis was placed on the following aspects of quasi-3D numerical modelling: overall computational efficiency of the model; the vertical eddy viscosity; the effects of nonlinear advective terms; the approximation of the boundary conditions; and coupling between the solution of the flow variables in the horizontal plane and that in the vertical direction. Vertical density stratification is not considered, although horizontal density gradients could be implemented.

With the aim to apply the flow model DUTRID to solve practical problems, a dispersion model for passive dissolved substances was also developed. Considering its advantages (see chapter two), the particle tracking method was chosen.

In this chapter the Q3D numerical model DUTRID is presented in detail. In the following section the main features of the model are described. The basic shallow water equations used in the model are derived in section 3.3. The main features of the depth-integrated flow module, which is based on an existing flow model, are summarized in section 3.4. A discussion about the advective terms through a 2D vorticity analysis is also presented in this section. The second module in DUTRID, namely, the velocity profile module, is described in section 3.5. The finite-difference approximations to the basic equations used in the module as well as to the boundary conditions are presented in detail. The computation procedure and coupling of the modules are described in section 3.6. Some conclusions and a summary of this chapter are given in section 3.7. The vertical eddy viscosity, which is a
most important parameter in shallow water flow models, will be discussed in the next chapter.

3.2 Main features of DUTRID

The basic feature of the Q3D flow model DUTRID is the application of the Vertical-Horizontal Splitting (VHS) algorithm. For one time step of computation, the flow velocities are first computed as functions of only the horizontal coordinates by using a 2DH flow module (which henceforth will be called depth-averaged flow module), then as functions of the vertical coordinate in the second module (velocity profile module). The model thus consists of two individual modules. An existing 2DH flow model DUCHESS (N. Booij, 1989) was used as the first module after some modifications.

One of the main feature of DUTRID is the possibility to couple the two modules. The depth-averaged flow module uses the bottom shear stresses computed in the velocity profile module, which, in turn, uses the free surface gradients that are computed in the first module. In this way, the formulation for the bottom shear stress in the depth-averaged module is improved. The last module can also be used as an independent model, but then the bottom shear stresses are related to the depth-averaged flow velocities, which is known to be a rather poor approximation for flows with noticeable circulations in the vertical plane, such as wind-induced flows (Hunter, 1975).

The bottom shear stress is computed by using the law of the wall and by introducing an equivalent roughness height. Each time new vertical profiles of horizontal velocities are computed in the second module, the bottom shear stresses are computed and used in the following step(s) of depth-averaged computations.

Adective terms in the equations of motion are usually neglected in Q3D flow models (see the preceding chapter). Though usually small under shallow water conditions, these terms can be shown to have marked effects on the flow patterns because of their cumulative effects. Therefore, they are retained in the basic formulation of DUTRID.
Another feature of DUTRID is the simple approach used to model the vertical eddy viscosity. It can be prescribed by the user as a constant value or as a given distribution in the vertical. Since the model was conceived a priori to be an efficient tool for practical applications, the incorporation of more sophisticated turbulence models such as the $k-\varepsilon$ model in the model, was not considered. Instead, a simple formulation was derived which includes the local wind shear stress, the bottom friction and the vertical coordinate. Among various alternatives, this formulation gives the best fit of computed velocity profiles to existing laboratory measurements in a wind-water flume.

Large gradients of flow velocities occur in the vicinity of the bottom and the free water surface exposed to wind shear. Although an equidistant grid was used in the vertical direction, only a relatively small number of grid points are needed in DUTRID for predicting the velocity profiles with high accuracy. This is due to proper approximations to the free surface and bottom boundary conditions. It is a particularly important feature since a large number of grid points in the vertical would result in a drastic increase in the computational cost.

**Particle Tracking Model and DUTRID**

The dispersion model developed in the present study runs in conjunction with the flow model DUTRID. Only passive dissolved matters were considered with the aim to concentrate on the two main dispersion processes, namely, the advective transport and the transport through turbulence. It is possible to extend the model to take account of other characteristics of a substance, such as erosion or sedimentation of solid particles.

The advection of particles in a given flow field is resolved analytically per grid cell. Net drift of particles which always arises when a numerical integration scheme is used, is thereby reduced. It can be shown that using this cell-analytical approach, the computed paths of particles remain closed in purely advective flow in a closed water body, provided the flow is divergence-free. This condition is automatically satisfied when the equation of continuity is used to solve the flow velocity distribution.

The transport of particles through turbulence is modelled through random displacements
related to the turbulent eddy viscosities, which are superimposed on the advective displacements. The effects of anisotropy of turbulence are taken into account.

3.3 The shallow water equations

3.3.1 Introduction

Shallow water conditions imply that the length scale of the water body in the vertical direction \(D\) is much smaller than the smaller one of length scales in the horizontal directions \(L\), or

\[
\delta = \frac{D}{L} \ll 1 .
\]  \hspace{1cm} (3-1)

By using this assumption the basic flow equations can be simplified following a normalization procedure (Stoker, 1957). A hydrostatic pressure distribution is obtained in first approximation. The simplified equations, usually called shallow water equations, are used in the flow model DUTRID.

3.3.2 The hydrostatic pressure distribution

The hydrostatic pressure distribution applied in shallow water flow modelling can be written as follows

\[
p = \rho g (H_z - z) + P ,
\]  \hspace{1cm} (3-2)

where \(p\) = hydraulic pressure; \(\rho\) = water density; \(g\) = acceleration due to gravity; \(H_z\) = level of the free water surface; \(z\) = the vertical coordinate (zero at a horizontal reference plane and positive upward). \(z = H_b\) at the bottom so that the water depth \(D = H_z - H_b\); \(P\) = atmospheric pressure.

Equation (3-2) is essential for Q3D flow modelling. The horizontal gradients of hydraulic pressure in the equations of momentum in the horizontal directions can thereby be expressed as the gradients of the free surface elevation, which makes it possible to apply the VHS algorithm.
Equation (3-2) can be derived from the basic turbulent flow equations by using equation (3-1). The derivation is given in the next section.

3.3.3 The shallow water equations

The shallow water equations can be derived from the basic turbulent flow equations written below.

The equation of continuity is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0,$$  \hspace{1cm} (3-3)

The equations of motion are

$$\frac{\partial u}{\partial t} + \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} + \frac{\partial (uw)}{\partial z} + \frac{\partial u'w'}{\partial x} + \frac{\partial u'v'}{\partial y} + \frac{\partial u'w'}{\partial z} - C_o v = -\frac{1}{\rho} \frac{\partial p}{\partial x},$$ \hspace{1cm} (3-4)

$$\frac{\partial v}{\partial t} + \frac{\partial (vu)}{\partial x} + \frac{\partial (vv)}{\partial y} + \frac{\partial (vw)}{\partial z} + \frac{\partial v'u'}{\partial x} + \frac{\partial v'v'}{\partial y} + \frac{\partial v'w'}{\partial z} + C_o u = -\frac{1}{\rho} \frac{\partial p}{\partial y},$$ \hspace{1cm} (3-5)

$$\frac{\partial w}{\partial t} + \frac{\partial (wu)}{\partial x} + \frac{\partial (wv)}{\partial y} + \frac{\partial (ww)}{\partial z} + \frac{\partial w'u'}{\partial x} + \frac{\partial w'v'}{\partial y} + \frac{\partial w'w'}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} - g,$$ \hspace{1cm} (3-6)

where \( u, v, \) and \( w \) = components of the ensemble mean flow velocity in \( x-, y- \) and \( z- \) directions; \( u', v' \) and \( w' \) = the corresponding turbulent components; \( C_o \) = the Coriolis parameter \( (C_o = 2\Omega \sin \varphi \) with \( \Omega = \) angular speed of the earth and \( \varphi = \) latitude). The molecular stresses in the equations of motion are neglected.

Equations (3-3) through (3-6) can be simplified following a normalization procedure and by using equation (3-1). The following normalized quantities are introduced:

$$\tilde{x} = \frac{x}{L}, \quad \tilde{y} = \frac{y}{L}, \quad \tilde{z} = \frac{z}{D};$$

$$\tilde{u} = \frac{u}{U'}, \quad \tilde{v} = \frac{v}{U'}, \quad \tilde{w} = \frac{w}{W'};$$

where \( U, V \) and \( W = \) characteristic flow velocity components in \( x-, y- \) and \( z- \) directions, respectively.

We further assume that the wind shear stresses exerted on the free water surface are the main driving force of the fluid flow. It is then reasonable to scale all the turbulent
exchanges terms such as $\bar{u}'u'$, $\bar{u}'v'$, $\bar{u}'w'$ etc. with $|W|/\rho$, $W$ being a characteristic wind shear stress vector. The hydraulic pressure can be scaled as
\[ p \sim \rho g D. \]

Introducing the normalized quantities into the continuity equation (3-3) yields:
\[ \frac{U \partial \bar{u}}{L \partial \bar{x}} + \frac{U \partial \bar{v}}{L \partial \bar{y}} + \frac{W \partial \bar{w}}{D \partial \bar{z}} = 0, \]
or, dropping the tilde (\~)
\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{WL}{DU} \frac{\partial w}{\partial z} = 0. \]

This equation holds provided that:
\[ \frac{WL}{UD} = 1, \text{ or } W = \frac{D}{L} U. \]

Therefore,
\[ W = \delta U \ll U. \quad (3-7) \]

Equation (3-7) shows that the vertical component of the flow velocities is quite small compared to the horizontal components. This is in fact typical for shallow water flows.

Rewritten in the original form, the continuity equation looks unchanged.

The hydrostatic pressure distribution law (equation (3-2)) can be derived by simplifying the equation of motion in the vertical direction (equation (3-6)). The turbulent exchange terms can be scaled as follows
\[ \frac{\partial \bar{w}'u'}{\partial x} \sim \frac{1}{L} \frac{|W|}{\rho}, \quad \frac{\partial \bar{w}'v'}{\partial y} \sim \frac{1}{L} \frac{|W|}{\rho}, \quad \frac{\partial \bar{w}'w'}{\partial z} \sim \frac{1}{D} \frac{|W|}{\rho}. \quad (3-8) \]

Since $D \ll L$, the first two terms related to horizontal turbulent exchanges are quite small compared to the third term. They can therefore be neglected in equation (3-6) normalized as follows:
\[ \frac{W \partial w}{T \partial t} + \frac{UW}{L} \left( \frac{\partial (wu)}{\partial x} + \frac{\partial (vw)}{\partial y} + \frac{\partial (ww)}{\partial z} \right) + \frac{|W|}{\rho D} \frac{\partial \bar{w}'w'}{\partial z} + g \frac{\partial p}{\partial z} + g = 0. \quad (3-9) \]

where $T$ is a characteristic time. For slowly varying boundary conditions it can be equal to either of the following two time scales:
* the long wave period \( T_1 = L/\sqrt{gD} \);

* the relaxation time \( T_2 = \rho UD/|W| \).

Dividing by \( g \) and substituting \( W = \delta U \), (3-9) becomes:

\[
\frac{\delta U}{gT} \frac{\partial w}{\partial t} + \frac{\delta U^2}{gL} \left[ \frac{\partial (wu)}{\partial x} + \frac{\partial (vw)}{\partial y} + \frac{\partial (ww)}{\partial z} \right] + \left| W \right| \frac{\partial w w'}{\partial z} + \frac{\partial p}{\partial z} + 1 = 0. \tag{3-10}
\]

Under normal wind conditions, \( |W|/(\rho g D) \ll 1 \). The vertical diffusion term is therefore negligible. Furthermore, \( \delta U/(g T) = \delta^2 Fr \) (if \( T = T_1 \)) where \( Fr \) is the Froude number \( (Fr = U/\sqrt{gD}) \), or \( \delta |W|/(\rho g D) \) (if \( T = T_2 \)). Under normal conditions of wind-driven flow, \( Fr \ll 1 \) so that in both cases \( \delta U/(g T) \ll 1 \). In other words, the (local) vertical acceleration of fluid flow is much smaller than \( g \).

As to the advective terms,

\[
\frac{\delta U^2}{gL} = \delta^2 Fr^2 \ll 1.
\]

Neglecting higher order terms, equation (3-10) reduces to \( \frac{\partial p}{\partial z} + 1 = 0 \), or \( \frac{1}{\rho g} \frac{\partial p}{\partial z} + 1 = 0 \) written in dimensional form. Integrating this equation over the water depth and using \( p = P \) at \( z = H \), one obtains equation (3-2).

It is useful to examine the relative importance of various terms in the equations of momentum in the horizontal directions (equations (3-4) and (3-5)). Examine first the turbulent exchange terms. Similar approximations can be made as for the equation in \( z \)-direction (c.f. equations (3-8)). The exchange terms in equation (3-4), for example, are:

\[
\frac{\partial u u'}{\partial x}, \quad \frac{\partial u v'}{\partial y}, \quad \frac{\partial u w'}{\partial z}.
\]

The first two terms are related to horizontal gradients of turbulent shear stresses of fluid. They are therefore small compared to the third term for shallow water flow.

Examine subsequently the relative importance of the advective terms and the Coriolis term in equations (3-4) and (3-5). Substituting from equations (3-7) and the hydrostatic pressure distribution law, the normalized equation (3-4) becomes:
\[
\frac{U}{T} \frac{\partial u}{\partial t} + \frac{UU}{L} \left( \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} + \frac{\partial (uw)}{\partial z} \right) + \frac{|W|}{\rho D} \left( \frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial y} + \frac{\partial w^*}{\partial z} \right) + C_a U v + g \frac{D}{L} \frac{\partial p}{\partial x} = 0. 
\] (3-11)

The factor multiplying each normalized quantity represents the order of magnitude of the term concerned. The order of magnitude (\(\gamma\)) of the advective terms compared with that of the dominant Reynolds stresses is therefore equal to

\[
\gamma = \frac{UU}{L} \frac{|W|}{\rho D} , \quad \text{or} \quad \gamma = \delta \frac{\rho U^2}{|W|} = \delta \frac{\rho}{\rho_a} \left( \frac{U}{u^*_a} \right)^2 ,
\] (3-12)

where \(u^*_a\) = wind friction velocity defined as follows

\[
u^*_a = \left( \frac{|W|}{\rho_a} \right)^{1/3}
\]

where \(\rho_a = \text{air density}\). One may choose as the velocity scale \(U\) a typical depth-averaged velocity which can generally be assumed to be equal to a fraction (say, one third) of a typical flow velocity near the free surface \(U_s\). Experimental evidence showed that (Wu, 1975) \(U_s = 0.55 u^*_a\), so that \(U \approx 0.2 u^*_a\). Introducing in addition the following typical values

\[
D = 10 m, \quad L = 10,000 m, \quad \rho = 1025 kg/m^3, \quad \rho_a = 1.225 kg/m^3
\]

one obtains \(\gamma \approx 0.04\). In other words, the advective terms seem to be small. However, these terms are nevertheless retained in the model equations of DUTRID because of their cumulative influences on flows, which will be analyzed in the next section.

The representative magnitude of the Coriolis term compared with the advective terms is given by the ratio below

\[
\frac{C_o U}{L} = \frac{U U}{L}, \quad \text{or} \quad \frac{1}{Ro},
\]

where \(Ro\) is the Rossby number. Introducing the following typical values

\[
C_o = 10^{-4} s^{-1}, \quad U = 0.1 m/s, \quad L = 10,000 m
\]

we have \(Ro = 0.1\). In other words, the Coriolis terms are approximately one order of magnitude larger than the advective terms. Considering the estimate of the advective terms given above, these terms may be significant in the model equations.
3.4 The depth-integrated flow

3.4.1 The depth-integrated flow module

An existing 2DH model called DUCHESS was used as the depth-integrated flow module in DUTRID after some modifications. DUCHESS can be used to compute storm surges and tides in shallow waters (N. Booij, 1989). General features of this model are outlined below; see N. Booij (1989) for further detail.

The basic flow equations used in DUCHESS are the depth-integrated equation of continuity and the equations of momentum in x- and y-directions.

The equation of continuity reads

$$\frac{\partial H_x}{\partial t} + \frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} = 0 .$$

(3-13)

The equation of momentum in x-direction is

$$\frac{\partial Q_x}{\partial t} + \frac{\partial}{\partial x} \left[ \frac{Q_x^2}{D} \right] + \frac{\partial}{\partial y} \left[ \frac{Q_x Q_y}{D} \right] - \frac{\partial}{\partial x} \left[ DE \frac{\partial}{\partial x} \left( \frac{Q_x}{D} \right) \right] -$$

$$\frac{\partial}{\partial y} \left[ DE \frac{\partial}{\partial y} \left( \frac{Q_x}{D} \right) \right] + gD \frac{\partial}{\partial x} (H_x + P) + Fr \frac{|Q_x|}{D^2} - C_v Q_x - \frac{W_x}{\rho} = 0 .$$

(3-14a)

in which \(Q\) = flow rate per unit width (vector); \(D\) = water-depth; \(E\) = horizontal eddy viscosity; \(Fr\) = friction coefficient. The subscripts \(x\) and \(y\) indicate components in the corresponding directions.

The equation of momentum in y-direction (equation (3-14b)) is analogous. It can be obtained by exchanging \(x\) with \(y\) (both differentials and subscripts) and replacing \(C_v\) by \(-C_v\) in the equation above.

These equations can be derived by integrating over the water depth the full 3D turbulent flow equations (3-3) through (3-5) in which the pressure is substituted from (3-2).

The following assumptions are used in this procedure:

(1) The horizontal turbulent exchange terms are modelled by using an empirical eddy viscosity coefficient \((E)\):
\[ \int_{h_i}^{h_f} \overline{u'w'} \, dz = -DE \frac{\partial \overline{u}}{\partial x} ; \quad \int_{h_i}^{h_f} \overline{uv} \, dz = -DE \frac{\partial \overline{v}}{\partial y} . \]

(2) The bottom shear stresses are expressed as quadratic functions of the depth-integrated flow velocities.

(3) The differential advection terms are neglected. These terms, some of which are defined below, are related to the vertical profiles of flow velocities:

\[ DC_{xx} = \frac{\partial}{\partial x} \int_{h_i}^{h_f} (u - \overline{u})^2 \, dz , \quad DC_{xy} = \frac{\partial}{\partial y} \int_{h_i}^{h_f} (u - \overline{u})(v - \overline{v}) \, dz \]

where a horizontal bar denotes depth-averaged quantities. Since they represent horizontal gradients of relatively small quantities, these terms are likely to be negligible for shallow water flows.

There are still uncertainties about the value of the eddy viscosity \( E \). Since the horizontal grid spacing is usually large in a shallow water flow model, the sub-grid-scale turbulence, which is not exactly known, should be represented by means of this viscosity. In DUCHESS, \( E \) can be given constant values or specified as a function of the local mean flow rate and the water-depth.

The above set of depth-integrated flow equations is solved by using the finite-difference method. A staggered equidistant grid in the horizontal plane is used (see figure 3.1). The approximation is central in space and approximately central in time. The boundary conditions are related either to the free surface elevation or to the flow rates. An Alternating Direction Implicit (ADI) Method is used to solve the equations for each time step. This numerical scheme is unconditionally stable.

The most essential modification made in DUCHESS used as a module in DUTRID, concerns the formulation of the bottom shear stresses. Originally in DUCHESS the bottom shear stresses are expressed as functions of the depth-integrated flow velocities. In DUTRID they are computed in the velocity profile module by using the law of the wall and by introducing an equivalent roughness height (see section 3.6).

The second modification is the inclusion of the differential advection terms in the depth-
averaged equations. With the computed velocity profiles these terms can be easily estimated in DUTRID.

3.4.2 The effect of advective terms

It was shown in section 3.3.3 that advective terms in the equations of momentum are small for wind-driven shallow water flows. They are therefore neglected in some Q3D flow models (see, for example, Lardner and Cekirge (1988)). In DUTRID, however, they are retained in the model equations for two reasons:

Firstly, they can be large in part of a flow field where the bottom slope is relatively large, and near an inflow or a discharge point. These effects are nevertheless usually confined to the vicinity of such locations so that the overall flow pattern in the flow field is little affected. Note that the present flow model is designed primarily for simulating large-scale flows. The basic shallow water assumption used in the model formulation, that is, the hydrostatic pressure distribution law, may not be satisfied at such locations.

Secondly, the advective terms are likely to have a cumulative influence on flows. They play a key role in the damping or dissipation process of vorticity in fluid flows, as is shown below through a two-dimensional vorticity analysis (Kranenburg, personal communication).

The following assumptions are made in this analysis:

* rigid-lid approximation;
* steady-state;
* horizontal turbulent exchanges are negligible.

Using these assumptions, the basic depth-integrated flow equations (3-13) and (3-14) can be simplified as follows:

\[
\begin{align*}
\left[ u \frac{\partial D}{\partial x} + \bar{v} \frac{\partial D}{\partial y} \right] + D \left[ \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right] &= 0 , \\
\eta \left[ u \frac{\partial u}{\partial x} + \bar{v} \frac{\partial u}{\partial y} \right] + g \frac{\partial H_z}{\partial x} - C_o \bar{v} &= S_x , \tag{3-16} \\
\eta \left[ u \frac{\partial v}{\partial x} + \bar{v} \frac{\partial v}{\partial y} \right] + g \frac{\partial H_z}{\partial y} + C_o \bar{u} &= S_y , \tag{3-17}
\end{align*}
\]

where

\[
S_x = \frac{1}{\rho D} (W_x - T_x) , \quad S_y = \frac{1}{\rho D} (W_y - T_y) ,
\]

and \( \eta = 1 \) (advective terms taken into account) or 0 (advective terms neglected). An overbar denotes depth averaged quantities.

Introducing the vorticity \( \xi = \partial \bar{v}/\partial x - \partial \bar{u}/\partial y \), we can derive the following relation by eliminating the pressure gradient terms between equations (3-16) and (3-17):

\[
\eta \frac{d}{dt} \left( \frac{C_o + \xi}{D} \right) = \frac{\partial S_y}{\partial x} - \frac{\partial S_x}{\partial y} , \tag{3-18}
\]

where \( d/dt = \bar{u} \partial/\partial x + \bar{v} \partial/\partial y \) is the material derivative. Assuming further:

* constant surface wind shear stresses;
* linear bottom friction resistance, or \( T_x = \rho k U \bar{u} \); \( T_y = \rho k U \bar{v} \) where \( U \) = reference flow velocity; \( k \) = friction coefficient,

equation (3-18) becomes:

\[
\eta \frac{d}{dt} \left( \frac{\xi + C_o}{D} \right) + \frac{kU}{D} \left( \frac{\xi + C_o}{D} \right) = \frac{kU C_o}{D} \\
+ \frac{1}{D^3} \left[ \left( kU \bar{v} - \frac{W_y}{\rho} \right) \frac{\partial D}{\partial x} + \left( kU \bar{u} - \frac{W_x}{\rho} \right) \frac{\partial D}{\partial y} \right] . \tag{3-19}
\]
Equation (3-19) describes the damping or decaying process of vorticity through the bottom friction resistance. A typical timescale for the decay of vorticity is:

\[ T_d = \frac{\eta D}{kU} \]

The related decay length is therefore

\[ L_d = U T_d = \frac{\eta D}{k} \]

One can see that vorticity is instantaneously dissipated (\( L_d = 0 \)) if the advective terms are neglected (\( \eta = 0 \)). If the advective terms are included in the equations, a large dissipation length can be expected. For example, assuming \( D = 10 \text{ m} \) and \( k = 0.004 \) (Simons et al, 1986), we have \( L_d = 2500 \text{ m} \).

Since the vorticity is coupled with the flow velocity through the stream function, the dissipation length \( L_d \) can also be interpreted as the length scale of the distortion of the flow caused by neglecting the advective terms. Disregarding the advective terms will obviously not influence the computed results of velocity distributions when the used grid spacing is larger than the dissipation length \( L_d \). In other words, this distortion can be observed in the numerical results only when the used grid spacing is much smaller than \( L_d \).

For the finite-difference approximations of the advective terms used in DUTRID one is referred to section 3.5.4.

### 3.5 The velocity profile module

#### 3.5.1 Basic equations

In the second module of DUTRID the vertical distributions of the velocity components are determined. The model equations used in this module are equivalent to equations (3-2) through (3-5), except that the horizontal diffusion terms are neglected.

The horizontal diffusion terms are neglected in the vertical profile module so that the vertical-horizontal splitting algorithm can conveniently be applied. For shallow water flow these terms are usually small compared to the vertical diffusion terms (see section 3.3.3). It
is therefore expected that they have little influence upon the vertical distributions of flow velocities.

It is noted that this approximation may give rise to small deviations between the local depth-averaged velocities computed in the two modules. This is especially the case near a lateral mixing zone or a boundary in a flow field where the horizontal exchange terms are locally important. These deviations may accumulate as computations proceed in time because of coupling of the two modules. A procedure of correction of the depth-averaged velocities is therefore introduced in DUTRID (see section 3.6).

The frequently applied Boussinesq assumption was used to model the horizontal turbulent shear stresses in equations (3-4) and (3-5):

$$\frac{\tau_x}{\rho} = -u'w' = \nu \frac{\partial u}{\partial z}, \quad \frac{\tau_y}{\rho} = -v'w' = \nu \frac{\partial v}{\partial z},$$

where $\nu$ = vertical eddy viscosity coefficient. This coefficient is assumed a function of local wind shear stress, the bottom shear stress and position in the flow field (see next chapter).

Summarizing, the basic equations used in this module are as follows. The equation of continuity is (equation (3-3)):

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0,$$

The equations of motion are:

$$\frac{\partial u}{\partial t} - \frac{\partial}{\partial z} \left( \nu \frac{\partial u}{\partial z} \right) - C_o \nu = STX,$$

$$\frac{\partial v}{\partial t} - \frac{\partial}{\partial z} \left( \nu \frac{\partial v}{\partial z} \right) + C_o u = STY,$$

where

$$STX = - \left( \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} + \frac{\partial (uw)}{\partial z} \right) - g \frac{\partial H_s}{\partial x},$$

$$STY = - \left( \frac{\partial (vu)}{\partial x} + \frac{\partial (vv)}{\partial y} + \frac{\partial (vw)}{\partial z} \right) - g \frac{\partial H_s}{\partial y}.$$

The nonlinear terms in equations (3-21) and (3-22) are deliberately moved to the right-
hand side of the equations. As will be seen later, they are approximated by using an explicit scheme in the finite-difference equations. Therefore, they are assumed known when the equations are solved for the velocity profiles.

Since the free water surface elevation is solved in the depth-integrated flow module, all the right-hand terms in equations (3-21) and (3-22) are known. These equations form a set of parabolic partial differential equations for $u$ and $v$ as functions of the vertical coordinate $z$ and time $t$. They can be solved by using finite-difference approximations (see section 3.5.3).

The equation of continuity (3-3) is used in the second module to compute the vertical velocity component $w$.

3.5.2 Boundary conditions

When solving equations (3-21) and (3-22) for the velocity profiles, boundary conditions must be specified at the free water surface and the bottom. Shallow water flows are essentially boundary layer flows characterized by large gradients of flow velocities near the boundaries. One could consider using a fine grid, or a non-equidistant grid, to model high velocity gradients near the boundaries. In DUTRID, however, only a small number of grid points in the vertical is allowed because of the computational efficiency of the model pursued.

In relation with the high velocity gradients, the vertical eddy viscosity approaches very small values near the bottom as well as the free surface (in case the surface wind shear is not too large). Both large velocity gradients and small values of the eddy viscosity near the boundaries could give rise to numerical difficulties when solving the equations. The approximations of the boundary conditions are therefore of particular importance from a numerical point of view. Also, the approach should be physically realistic.

The bottom boundary

Several alternatives of the approximation to the bottom boundary condition exist. The following four are among the most frequently used.
1) Free slip

Free slip means that the bottom surface does not resist the fluid flow. The bottom shear stress is therefore equal to zero. Obviously, this is the most simplified approximation of the bottom boundary condition. This formulation is often used in semi-analytical Q3D models in which, with a simple formulation of the vertical eddy viscosity, the vertical profiles of velocities are resolved analytically.

2) Shear stress depending on mean velocity, \( T = \rho F_r |U|U \), where \( T \) = bottom shear stress vector; \( U \) = depth-averaged flow velocity vector; \( F_r \) = friction coefficient. This is a useful approximation for modelling vertically uniform flows, such as open channel flows. It is used in most 2DH flow models (also in DUCHESS).

However, this approximation is physically uncertain when used for modelling shallow water flows. The bottom shear stresses arise because of the local gradients of flow velocities near the bottom. Therefore, they should be related to the velocity field near the bottom rather than the depth-averaged flow velocities. Since the velocity vector may change in both magnitude and orientation in the vertical direction, especially in wind-induced flows in closed water bodies, this approximation is inappropriate for modelling the flows under consideration.

3) Shear stress depending on near-bed velocity or near-bed velocity gradient,

\[
T = \rho F_r |U_b|U_b, \quad \text{or} \quad T = \rho \left( \nu \frac{\partial U}{\partial z} \right)_{z=h_a+z_c},
\]

where \( U_b \) = local flow velocity near the bottom. This is a physically realistic approximation as is evident from the discussion above. However, the definition of \( U_b \) is somewhat arbitrary. The implementation of the second equation in the numerical model may result in numerical difficulties because the eddy viscosity approaches zero near the bottom.

4) No-slip

In shallow water flows there is a thin layer of approximately constant shear stress near the bottom. In this constant-stress layer, a logarithmic velocity distribution can be assumed:
\[ u = \frac{u_*}{\kappa} \ln \left( \frac{z-H_b}{z_0} \right) , \]  

where \( u_* \) = reference friction velocity related to the bottom shear stress (see section 3.6.2); \( \kappa = \text{Von Karman coefficient} \); \( z_0 \) = equivalent bottom roughness height. \( z_0 \) should be prescribed at every grid point in the horizontal plane. Equation (3-23) is usually called law of the wall.

In general \( z_0 \) is a function of the local flow velocity and the physical bottom roughness. In practical situations, however, the bottom is nearly always hydraulically rough so that \( z_0 \) only depends on the bottom geometry. The velocity components \( u \) and \( v \) vanish (no-slip) at \( z = H_b + z_0 \), as is evident from equation (3-23).

The last one of the four alternatives presented above is used in DUTRID because it is physically correct. Furthermore, numerical difficulties are avoided since it does not directly involve the vertical eddy viscosity.

The free surface boundary

Several approximations to the free surface boundary condition exist. The following three are among the most frequently used. The third approximation is used in DUTRID.

1) **Shear stress depending on near-surface velocity**, 
\[ \rho C_f |U_s| U_s = W , \]
where \( U_s \) = flow velocity near the surface; \( C_f \) = empirical coefficient.

This empirical relation postulates that the flow velocity near the free water surface depends only on the wind shear stresses, and that this velocity is in the wind direction. It does not take into account the influence of other factors such as the pressure gradient, the Coriolis force and the vertical momentum transfer. Furthermore, there are uncertainties about the values to be assigned to the empirical coefficient \( C_f \).

2) **Shear stress proportional to the near-surface velocity gradient**, 
\[ \rho \nu \frac{\partial U}{\partial z} = W , \quad z = H_s . \]
In the discretized equations the velocity gradient in the above expression is replaced by a 
finite-difference approximation.

This relation implies that the wind shear stresses at the surface are balanced by the 
local turbulent shear stresses in the fluid flow, which is physically correct. However, its 
implementation in the numerical model may result in numerical difficulties when the eddy 
viscosity approaches zero near the boundary. This could be the case when little turbulence 
is generated near the free surface by wind waves. When wind waves are present and the local 
vertical eddy viscosity is no longer small, this relation may be useful.

3) **Near-surface shear stress equal to imposed wind shear stress**, 

\[ \tau = W, \quad z = H_z, \]

where \( \tau \) is turbulent shear stress. Near the free surface the vertical diffusion terms in the 
basic equations are written as gradients of turbulent shear stresses (see equation (3-20)). 
These gradients are approximated by finite-differences in which, according to the above 
equation, the shear stress at the free surface can be substituted by the wind shear stress. A 
special arrangement of the finite-difference grid in the vertical direction is needed for that 
purpose (see next section).

This relation has the same physical meaning as the preceding alternative. However, 
since neither the eddy viscosity nor the velocity gradient are involved in the finite-difference 
approximations, numerical difficulties are avoided.

**Lateral boundaries**

Lateral boundary conditions are approximated in the depth-averaged module either as 
given discharges or as prescribed free surface elevations. In the velocity profile module, 
vertically uniform velocity profiles are assumed at lateral boundaries with given discharges, 
whereas at boundaries with prescribed free surface elevations the velocity profiles are 
computed since the local free surface gradients can then be approximated.

The above treatment of the lateral boundary conditions is a first approximation. One 
could consider using more accurate approximations. However, since the influence of a lateral 
boundary on a flow is limited to a region of the order of the water depth away from that
boundary, and the water depth is assumed much smaller than the horizontal dimensions of the flow field, such an effort is not justified.

3.5.3 Finite-difference equations

Equations (3-21) and (3-22) form a set of second order parabolic partial differential equations. They are solved in the second module by using finite-difference approximations.

The Crank-Nicholson scheme used for the heat equation (Ames, 1977) is applied. The computations are performed at each individual $H$ point in the staggered grid in the horizontal plane (see figure 3.1). The vertical grid spacing is equidistant and an equal number of grid points ($m$) is used for all verticals. The first vertical grid point is situated at $z = H_b + z_0$ where a no-slip condition is applied. The highest grid point is placed half a grid spacing below the free surface (see figure 3.2 below). For flows in lakes and reservoirs as considered in this study, the variations in the level of the free water surface are usually small. Temporal variations in the position of the vertical grid are therefore not considered.

A general implicit finite-difference approximation to the equation of motion in $x$-direction (equation (3-21)) can be written as follows.

$$\frac{u_{k+1}^n + u_k^n}{\Delta t} - \alpha \frac{1}{\Delta z} \left( \frac{\nu_{k+1}^n + \nu_k^n}{2} \frac{u_{k+1}^{n+1} - u_k^{n+1}}{\Delta z} - \frac{\nu_k^n + \nu_{k+1}^n}{2} \frac{u_k^{n+1} - u_{k+1}^{n+1}}{\Delta z} \right)$$

$$- (1 - \alpha) \frac{1}{\Delta z} \left( \frac{\nu_{k+1}^n + \nu_k^n}{2} \frac{u_{k+1}^n - u_k^n}{\Delta z} - \frac{\nu_k^n + \nu_{k-1}^n}{2} \frac{u_k^n - u_{k-1}^n}{\Delta z} \right)$$

(3-24)

$$- C_o \nu_k^n = STX_k^n,$$

where $k$ is the vertical level index of a grid point, ($k = 1, 2, \ldots, m$); $n$ = index of time level; $\Delta t$ = time-step; $\Delta z = $ vertical grid spacing; $\alpha = $ weighting coefficient. The source term $STX$ consists of the advective terms and a pressure gradient (see section 3.5.1). The approximations to the advective terms are given in the following section. The pressure gradients are computed in the 2DH module.

Equation (3-24) can be rearranged as follows
Figure 3.2. Vertical grid used in the velocity profile module.

\[ B_k u_{k-1}^{n+1} + C_k u_k^{n+1} + F_k u_{k+1}^{n+1} = P_k, \]  

(3-25)

where the coefficients \( B_k, C_k, F_k, \) and \( P_k \) consist of known quantities (coefficients and velocities at time level \( n \)). Writing this equation for all levels in a vertical, a linear tridiagonal system is obtained that can easily be solved for \( u_k^{n+1} \) \((k = 1, 2, \ldots, m)\).

The finite difference approximation to the equation of motion in \( y \)-direction (equation (3-22)) can easily be obtained by replacing \( u \) with \( v \), \( C_o \) with \( -C_o \) and \( STX \) with \( STY \).

In equations (3-21) and (3-22) the solutions of \( u \) and \( v \) are coupled through the Coriolis force. One should therefore solve these equations simultaneously. However, in case of a weak coupling, which could be encountered under certain flow conditions, numerical difficulties arise when solving the simultaneous algebraic equations. Therefore, these coupling terms are treated explicitly in the finite-difference approximations in DUTRID. This treatment also leads to a reduction in the number of simultaneous equations to be solved at each time-step, hence an increase in the efficiency of the model.

In case \( \alpha = 0.5 \), we have a Crank-Nicholson scheme which is unconditionally stable with constant coefficients. The accuracy of the scheme is \( O(\Delta z^2, \Delta t^2) \).
The continuity equation (3-3) is used to compute the vertical component of the flow velocity \( w \). Its finite-difference approximation can be written as

\[
\frac{u_{i+1} - u_{i-1}}{2 \Delta x} + \frac{v_{j+1} - v_{j-1}}{2 \Delta y} + \frac{w_{k+1/2} - w_{k-1/2}}{\Delta z} = 0.
\]

Note that the used grid is non-orthogonal in the vertical planes in case of an uneven bottom surface. Care should be taken as to the approximations of horizontal gradient terms in this equation (see the following section).

The finite-difference approximations to the boundary conditions are given below. Since they are quite similar for both horizontal directions, only the approximations in \( x \)-direction are given.

The bottom boundary condition

A no-slip condition is assumed at the grid point where \( z = H_b + z_0 \), or

\[
u_{i+1}^n = u_i^n = 0.
\]

Furthermore, a logarithmic velocity distribution is assumed between this grid point and the grid point above (see figure 3.2).

The free surface boundary condition

Equation (3-24) written for the first grid point under the free water surface \( k = m \) can be slightly modified by using equation (3-20):

\[
\frac{u_m^{n+1} + u_m^n}{\Delta t} = \alpha \frac{1}{\Delta z} \left( \frac{\tau_{ss}^{n+1}}{\rho} - \frac{\nu_m^n + \nu_{m-1}^n}{2} \frac{u_m^{n+1} - u_m^{n-1}}{\Delta z} \right) - (1 - \alpha) \frac{1}{\Delta z} \left( \frac{\tau_{ss}^n}{\rho} - \frac{\nu_m^n + \nu_{m-1}^n}{2} \frac{u_m^n - u_m^{n-1}}{\Delta z} \right) - C_o v_m^n = STX_m^n,
\]

where \( \tau_{ss} = W_s \).

The turbulent shear stress at the free surface is replaced by the wind shear stress. It is seen that the vertical eddy viscosity at the free surface, which may take a very small value, is not present in the equation.

Because of the configuration of the vertical grid, the flow velocity at the free water
surface is not computed in the flow model. The nearest computed flow velocity is located half a grid spacing lower. This velocity approaches the flow velocity at the free surface with an increasing number of nodes in the vertical grid. One could also use extrapolation by assuming, for example, a logarithmic distribution of the velocity between the free surface and the nearest grid point.

3.5.4 Numerical approximation of the advective terms

The advective terms in the equations of motion for the velocity profile module can be approximated only explicitly in order to apply the VHS algorithm. While higher order approximations are used for other terms in the equations (see the section above), these approximations have an accuracy of the first-order. However, since the magnitude of the advective terms is usually small (see section 3.3.3), the inaccuracy introduced by using the first-order approximations in the velocity profile module should be insignificant. Note that the vorticity analysis of section 3.4.2 is concerned with the depth-averaged flow.

The advective terms are approximated explicitly in the velocity profile module. In other words, at each time step of computation, they are computed by using the velocity profiles at the previous time-step. The first-order upstream scheme is used because of its numerical stability.

The advective terms are

\[ CT_x = u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z}, \quad CT_y = u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z}. \]

For a typical term \( u \frac{\partial u}{\partial x} \) containing velocity gradient in \( x \)-direction, the following finite-difference approximation in an orthogonal grid is used at the grid point \( (i, j, k) \):

\[
\frac{u}{\partial x} = \begin{cases} 
    u_{i,j,k} \frac{u_{i,j,k} - u_{i-1,j,k}}{\Delta x}, & u_{i,j,k} > 0; \\
    u_{i,j,k} \frac{u_{i+1,j,k} - u_{i,j,k}}{\Delta x}, & u_{i,j,k} < 0.
\end{cases}
\]

Approximations to other gradient terms in the expressions of the advective terms are similar. The approximation to \( CT_z \) is, therefore, for the case \( u, v, w > 0 \):

\[
0
\]
The grid used in DUTRID is orthogonal in the horizontal plane, and also in vertical planes in case of a horizontal bottom (see figure 3.1 and 3.2). In general, however, the bottom is uneven. Since an equal number of grid points is used for all verticals, the grid is then non-orthogonal in vertical planes (see figure 3.3 above). Corrections should be made to the finite-difference approximations of the horizontal gradient terms such as \( \frac{\partial u}{\partial x} \), \( \frac{\partial u}{\partial y} \), \( \frac{\partial v}{\partial x} \) and \( \frac{\partial v}{\partial y} \) based on values of velocities at nodal points.

Since the treatment is similar for all the horizontal gradients, we consider here a typical term, \( \frac{\partial u}{\partial x} \), case \( u, w < 0 \), as an example. We can make a first-order Taylor expansion of \( u \) at grid point \((i, j, k)\):

\[
\begin{align*}
\eta_{i-1,j,k} &= \eta_{i,j,k} + \frac{\partial u}{\partial x} \Delta x + \frac{\partial u}{\partial z}(z_{i+1,j,k} - z_{i,j,k}) \\
&
\end{align*}
\]

from which we can derive the following expression for \( \frac{\partial u}{\partial x} \) at point \((i, j, k)\):

\[
\frac{\partial u}{\partial x} \approx \frac{u_{i+1,j,k} - u_{i,j,k}}{\Delta x} - \frac{\partial u}{\partial z} \frac{z_{i+1,j,k} - z_{i,j,k}}{\Delta x},
\]

where

\[
CT_x \approx u_{i,j,k} \frac{u_{i+1,j,k} - u_{i,j,k}}{\Delta x} + v_{i,j,k} \frac{u_{i,j,k} - u_{i,j-1,k}}{\Delta y} + w_{i,j,k} \frac{u_{i,j,k} - u_{i,j,k-1}}{\Delta z}.
\]
\[
\frac{\partial u}{\partial z} = \frac{u_{i,j,k+1} - u_{i,j,k}}{\Delta z}.
\]

The central difference approximations of horizontal gradients \( \partial u/\partial x \), \( \partial v/\partial y \) in the equation of continuity (see section 3.5.3) also need to be corrected for the non-orthogonality in vertical planes. Using a first-order Taylor expansion of \( u \) at grid point \((i, j, k)\):

\[
u_{i-1,j,k} = u_{i,j,k} - \frac{\partial u}{\partial x} \Delta x + \frac{\partial u}{\partial z} (z_{i-1,j,k} - z_{i,j,k}),
\]

we have, together with the above expression for \( u_{i+1,j,k} \):

\[
\frac{\partial u}{\partial x} = \frac{u_{i+1,j,k} - u_{i-1,j,k}}{2\Delta x} - \frac{\partial u}{\partial z} \frac{z_{i-1,j,k} - z_{i,j,k}}{2\Delta x}.
\]

(3-29)

## 3.6 Computation procedure

### 3.6.1 Coupling of modules

The present flow model DUTRID consists of two separate modules, namely, the depth-averaged (2DH) module and the velocity profile module. For one time-step of computation, two steps of computation are made. Firstly, the depth-averaged flow computations are made in a two-dimensional horizontal grid. Then in the second module, the vertical profiles of the flow velocities are determined for each vertical in that grid.

The two modules are coupled through the free surface elevation \( H_f \) and the bottom shear stresses \( T \). The free surface elevation and its horizontal gradients are computed in the 2DH module. They are used as input to the second module in which the vertical profiles of horizontal velocities are computed, with which the bottom shear stresses are calculated. These stresses are then used in the next one or several step(s) of depth-averaged computations until new velocity profiles are computed. This computational procedure is sketched in figure 3.4. The velocity profiles are corrected such that the depth-integrated value of a velocity profile equals that computed in the 2DH module (see section 3.6.3).

It is sometimes possible to make several steps of depth-averaged computations in the
2DH module before new velocity profiles are computed. The efficiency of the model can thereby be increased. Using a larger time-step in the velocity profile module than in the depth-averaged module is allowed for gradually varying flow conditions. Physically, the velocity profile module mainly represents the vertical diffusion process of momentum. This process is usually much slower than the propagation of gravity waves that is represented in the first module. Furthermore, in case of slowly varying wind action, a long period is needed in order for this process to be completed after the waves are sufficiently damped through the bottom shear stress. The bottom shear stress, which is fed-back from the velocity profile module into the depth-averaged module, is likely to change very slowly during this period. A larger time-step in the velocity profile module is therefore justifiable in that case.

For rapidly changing flow conditions, such as during the starting stage of a flow or rapidly changing surface wind action, numerical stability of the 2DH module requires the same time-step to be used in both modules, because of the explicit approximation of the bottom shear stress used in the 2DH module.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{diagram.png}
\caption{Computation procedure in DUTRID.}
\end{figure}
3.6.2 Computation of the bottom shear stress

The bottom shear stresses $T$ are computed in the second module after the velocity profiles have been resolved. Following equation (3-20):

$$ T = -\rho \nu \frac{\partial u}{\partial z}, \quad z = H_b + z_0. \quad (3-30) $$

The vertical eddy viscosity $\nu$ is in general related to the wind shear stresses, the bottom shear stresses and the vertical coordinate (see chapter 4). Assuming a viscosity model that is local as far as the horizontal coordinates are concerned, we can write formally:

$$ \nu = f\left( |u_b^*|, |u_i^*|, D \right) F \left( \frac{z - H_b}{D} \right), $$

where $u_i^*$ = surface-friction velocity and $u_b^*$ = bed-friction velocity which are defined as follows

$$ T = -\rho |u_b^*| u_b^*; \quad W = \rho |u_i^*| u_i^*. $$

To compute the velocity gradient near the bottom in equation (3-30), a logarithmic distribution of the velocity is assumed between the no-slip grid point (where $z = H_b + z_0$) and the grid point above (where $z = H_b + z_1$, see figure 3.2), or

$$ u = \frac{u_0^*}{\kappa} \ln \frac{z - H_b}{z_0}, \quad z_0 \leq z - H_b \leq z_1 \quad (3-31) $$

according to which

$$ \frac{\partial u}{\partial z} = \frac{u_0^*}{\kappa z_0} \quad \text{at} \quad z - H_b = z_0. $$

According to equation (3-30), the bottom shear stress is, therefore:

$$ T = -\rho f\left( |u_b^*|, |u_i^*|, D \right) F \left( \frac{z_0}{D} \right) \frac{u_0^*}{\kappa z_0}, $$

or, inserting the definition of $u_b^*$ :

$$ |u_b^*| u_b^* = f\left( |u_b^*|, |u_i^*|, D \right) F \left( \frac{z_0}{D} \right) \frac{u_0^*}{\kappa z_0}. \quad (3-32) $$

In this equation $u_i^*$ and $z_0$ are prescribed quantities, and $u_0^*$ is related to the nodal velocity
\( u_1 \) at \( z = H_b + z_1 \) by equation (3-31), or

\[
    u_1 = \frac{u_0^*}{\kappa} \ln \frac{z_1}{z_0}.
\]

With \( u_0^* \) being the only unknown variable, equation (3-32) can be solved to obtain the bottom shear stress. Depending on the assumed function \( f \), iteration may be needed. A discussion on the vertical eddy viscosity will be presented in the next chapter.

3.6.3 Correction of velocity profiles

As one step in the coupling procedure, the depth-averaged velocities computed in the 2DH module are used to make corrections to the velocity profiles computed in the second module, such that the depth-averaged velocities in both modules are equal. The profiles are adjusted by adding at all points in the vertical a certain value (\( \Delta u \)) given by

\[
    \Delta u(x,y,t) \ D(x,y) = Q(x,y,t) - \int_{u_0}^{u_n} u(x,y,z,t) \ dz ,
\]

where \( Q(x,y,t) \) is obtained in the depth-averaged module.

Obviously, this value should theoretically be equal to zero. However, differences between the depth-averaged velocities computed in the two modules can occur on account of the (slight) inconsistency between the basic equations used in the two modules. The horizontal exchange terms are retained in the 2DH module, whereas they are neglected in the velocity profile module. Secondly, the different approximations in the two modules to various terms, such as the advective terms, may also contribute to these differences. Although the expected differences are small, they should be corrected in order to prevent accumulation through coupling of the two modules.

Some steady-state test computations using the flow model (see chapter 5) showed that this correction procedure has good numerical stability and little influence on the accuracy of the results. An alternative way of adjusting velocity profiles would be multiplication by a certain factor (close to unity). This alternative has not been adopted, however, because singularities may arise at certain locations in the flow field where the depth-integrated flow rate approaches zero.
3.7 Summary

A description of the Q3D numerical model DUTRID for shallow water flow was given in this chapter. The following aspects and features of the model were addressed: general features of the flow model; basic equations used in the two modules; main features of the 2DH module; differential equations and finite-difference approximations used in the velocity profile module; modelling of the free surface and bottom boundary conditions; influence of the advective terms; computational procedure and coupling of modules.

By using the shallow water assumption, a hydrostatic pressure distribution is obtained in first approximation. The VHS algorithm can then be applied in the flow model. The horizontal turbulent exchange terms are shown to be small. They are nevertheless retained in the depth-averaged module, but they are neglected in the velocity profile module. It is expected that this approximation has little influence upon the computed vertical distributions of velocities, and therefore on the depth-averaged flow velocities.

The two modules in DUTRID are coupled through the free surface elevation gradients and the bottom shear stresses. The bottom shear stresses are computed by using the velocity profiles obtained in the second module. As such, the formulation of the bottom shear stresses in the 2DH model used as a module in DUTRID is improved. The original formulation was based on the depth-averaged flow velocities.

The bottom boundary condition is modelled by using a no-slip condition and by introducing an equivalent roughness height. The turbulent shear stresses at the free surface in the discretized equations of motion are directly replaced by the imposed wind shear stresses. The eddy viscosities at the boundaries are therefore not involved in the finite-difference approximations. In this way, numerical difficulties that could arise from large velocity gradients and small vertical eddy viscosities at the bottom and the free water surface boundaries, are avoided.

The advective terms in the basic flow equations are shown to be small for shallow water flows with a small Froude number. Nevertheless, they may have noticeable consequences upon flows due to their cumulative effects, as was illustrated through a simple vorticity analysis. These terms are therefore retained in the equations used in both modules.
In the velocity profile module they are approximated explicitly by using a first-order upstream scheme. In the depth-averaged module a linearized implicit scheme is used in the computational direction, and an explicit higher order upstream scheme is used in the other direction (ADI).
CHAPTER 4

A STUDY OF THE VERTICAL EDDY VISCOSITY

4.1 Introduction

The turbulent shear stresses in the equations of momentum used in DUTRID are modelled by adopting the Boussinesq assumption (equation (3-20)). An eddy viscosity coefficient ($\nu$) for the vertical exchange of momentum, usually called "vertical eddy viscosity", was thereby introduced. As can be seen from the basic equations used in the velocity profile module (equations (3-21) and (3-22)), the eddy viscosity $\nu$ obviously has a crucial influence on the computed velocity profiles. Arbitrary distribution of this viscosity is allowed in the flow model. It can either be equal to a vertically constant value or be formulated as a function of vertical position, local water depth, bottom roughness, wind shear stress and flow conditions.

Considering the pursued computational efficiency of the flow model, it was not intended to use a higher-order turbulence model, such as the k-\$\epsilon$ model, which is costly in CPU time. Instead, a simple algebraic eddy viscosity model is used as the default option. This model is based on a numerical study presented in this chapter. In this study the flow model DUTRID, combined with various formulations of the eddy viscosity, is used to simulate velocity distributions in wind-driven flows measured by Yu (1987) in a wind-water flume. The resulting viscosity model takes into account the local bottom roughness, wind shear stress and flow conditions. Computations using a one-dimensional k-\$\epsilon$ model adapted for this particular case were also made to verify the eddy viscosity model.

In the next section of this chapter the experiments of Yu (1987) are described. Then
the computations using DUTRID simulating these experiments are shown. Alternative formulations of $\nu$ are considered and compared with the measurements of velocity profiles (section 4.3). Computations using the $k-\epsilon$ model, which supports the proposed formulation of $\nu$, are presented in section 4.4. Some conclusions and a discussion are presented in section 4.5.

4.2 The experiment of Yu (1987)

A systematic experimental study of turbulent flows induced by wind and a free surface gradient was made by Yu (1987). The experiment was carried out in a flume approximately 38 m long, with a cross section of 0.80 m $\times$ 0.59 m, see figure 4.1. The flume was provided with a roof, a suction ventilator at one end of the flume and an air intake at the other end such that a driving air current could be created. The velocity of the air current ($u_a$) measured at 10 cm above the water surface varied from 3.7 m/s to 8.0 m/s. The still water depth was kept at 20 cm. Steady water flows were created by using a pump, with cross-sectional average velocities ($\bar{u}_w$) varying from 4.75 cm/s to 21.4 cm/s. The flow in the water layer could be in a direction either following or opposing the air flow. The bottom of the

![Figure 4.1](image-url)  

*Figure 4.1. Sketch of wind-water flume used in experiments on wind-driven current (Yu, 1987). Lengths in cm.*
flume was made of cement. All flows tested were found to be in the hydraulically smooth regime (Yu, 1987).

The test section was located near the middle of the flume where the water depth was little affected by the wind set-up of the free surface. A laser-Doppler anemometer was used to measure the vertical distributions of the horizontal water and air flow velocities.

A total number of 24 combinations of air flow and water flow velocities were examined in the experiment. Using the measurements, Yu tested Reid’s (1957) general formulation of wind driven current profiles. He also proposed a velocity distribution composed of two logarithmic profiles, one for the flow near the free water surface, and the other for the flow near the bottom. This distribution was found to agree well with the measurements. Upon examining the two formulations of the velocity profiles, Yu obtained the values of the surface friction velocity \( u^* \) and the equivalent bottom roughness height \( z_0 \) by means of the profile method (see table 4.1).

4.3 Computations using DUTRID

The vertical eddy viscosity \( \nu \) is an important parameter in a 3D or Q3D numerical circulation model. Depending on the specific flow conditions to which a flow model was applied, various formulations of \( \nu \) were considered by different authors. Vertically uniform distributions were often used. For example, Kielmann and Kowalik (1980) assumed a constant eddy viscosity in their model of flows in shallow seas. Jamart and Ozer (1987) considered constant, exponential and linear distributions when modelling storm surges in shallow continental seas. Leendertse (1989) used vertically uniform distributions to simulate circulations induced by wind in Lake Yssel in the Netherlands.

The flow model DUTRID developed in the present study was designed to simulate shallow water flows driven mainly by wind shear and free surface gradient. It is well known that in such flows two boundary layers develop, one near the free surface, and the other near the bottom. When the water depth is of the order of several meters, as is the case with Lake Yssel (see chapter 7), these two boundary layers may overlap and interact with each other.
Accordingly, as far as the process of vertical momentum transfer is concerned, the flows under such circumstances are similar to those examined by Yu (1987). Computations were therefore made with DUTRID to simulate these flows, with the aim to find a suitable formulation of the vertical eddy viscosity.

Of the 24 combinations of the air and water flow velocities examined by Yu, eight typical experiments were selected and used in the present computations. These combinations are listed in Table 4.1, in which the corresponding values of surface-friction velocity and the bottom roughness height obtained by Yu are also given. In the computations these values were used as input to DUTRID. "Opposing wind" and "following wind" refer to the direction of the wind with respect to the cross-sectionally averaged water flow velocity.

Table 4.1. Experiments made by Yu (1987) that were simulated using DUTRID.

<table>
<thead>
<tr>
<th>No.</th>
<th>opposing wind</th>
<th>following wind</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( \bar{u}_w ) cm/s</td>
<td>( u_a ) m/s</td>
</tr>
<tr>
<td>1</td>
<td>4.75</td>
<td>5.7</td>
</tr>
<tr>
<td>2</td>
<td>10.4</td>
<td>5.7</td>
</tr>
<tr>
<td>3</td>
<td>14.1</td>
<td>5.7</td>
</tr>
<tr>
<td>4</td>
<td>18.0</td>
<td>5.7</td>
</tr>
</tbody>
</table>

The flume is modelled in all the numerical simulations using 12×2 grid points in the horizontal plane and 40 levels in the vertical. The horizontal grid spacing is 4 m. The water depth is 20 cm in still water. The channel width used in the computations was much larger than the actual size of the flume. But this has no consequences upon the velocity distributions as the side walls have negligible influence on the flow. Computation begins from still water with suddenly imposed wind shear, and terminates when a steady state is reached. A time-step of 5 s was used. The computed velocity profiles at the central point of the channel are
used in the analysis below.

Physically, the vertical eddy viscosity can be conceived as a variable depending on local flow conditions. Assuming horizontally uniform or slowly varying flow, it can be expressed as a function of the local wind shear stress exerted on the free surface, the bottom shear stress, the water depth and the vertical coordinate:

$$
\nu = f\left( |u_b^*|, |u_s^*|, D \right) F \left( \frac{z - H_b}{D} \right)
$$

(4-1)

where $u_s^*$ = surface-friction velocity; $u_b^*$ = bed-friction velocity.

As was mentioned above, shallow water flows can generally be considered as boundary layer flows. It is well known that satisfactory results for the vertical distribution of the flow velocity can be obtained using a parabolic distribution of $\nu$:

$$
\nu = \kappa |u^*| \left( z - H_b \right) \left( 1 - \frac{z - H_b}{D} \right)
$$

(4-2)

where $u^*$ = characteristic friction velocity. This velocity can be expressed as a function of the surface-friction velocity and the bed-friction velocity as below:

$$
|u^*| = f_1 \left( |u_b^*|, |u_s^*| \right).
$$

(4-3)

Computations using DUTRID were made for various expressions of the function $f_1$ in equation (4-3). The suitability of each expression was judged by comparing the corresponding computed velocity profiles with the measurements. The following cases were considered:

1. **Turbulence generated at the bottom boundary predominates,**

   $$
   |u^*| = |u_b^*|.
   $$

   (4-4)

   This relation is commonly used for open channel flows where pressure gradient is the dominant driving force. For some of the flows simulated in which wind shear dominates, the computed velocities using this relation show markedly larger gradients than the measurements, as can be seen in figures 4.2c. This suggests that equation (4-4) then **underestimates** $\nu$, and that when estimating the turbulence level in the water column, the
turbulence generated at the free surface boundary has to be taken into account.

(2) Linear superposition of friction velocities,

\[ |u^*| = |u_b^*| + |u_s^*|. \] (4-5)

The velocity profiles obtained by using this relation are generally much too uniform for all experiments simulated (see figure 4.2), which suggests that equation (4-5) overestimates the vertical eddy viscosity.

It is useful to consider the following well-known relations:

- \[ |u^*| = |u_b^*| \] when \( u_s^* = 0 \). This relation is applicable when the bottom boundary is the only source of turbulence, as is the case with open-channel flows.

- \[ |u^*| = |u_s^*| \] when \( u_b^* = 0 \). Considering the free surface boundary as a source of turbulence like the bottom boundary in the previous case, this relation is straightforward.

- \[ |u^*| = |u_b^*| = |u_s^*| \] when \( u_s^* = u_b^* \). This assumption gives the best fit for turbulent Couette flow between two parallel plates (Reichardt, 1959).

One could consider a general expression that interpolates between these three typical cases:

(3) Friction velocity depending on both sources of turbulence,

\[ |u^*|^n = |u_b^*|^n + |u_s^*|^n - \left( |u_b^*| |u_s^*| \right)^{\frac{n}{2}}, \quad n \geq 1. \] (4-6)

Some of the numerical results of the velocity profiles using equation (4-6) with \( n = 2 \) are shown in figure 4.2. Good agreement was found for both opposing and following wind actions. Computations with other values of \( n \) (1, 3 and 4) showed no better general agreement between computed velocity profiles and measurements.

(4) Friction velocity depending on the stronger source of turbulence,

\[ |u^*| = \max\left( |u_b^*|, |u_s^*| \right). \] (4-7)
Figure 4.2. Vertical profiles of horizontal velocities in a wind-water flume.
(a): case No. 1 in table 4.1; (b): case No. 4; (c): case No. 5.
○ measurement (Yu, 1987); computations with \( u^* \) according to (4.4) (- - -),
to (4-5) ( - - - ), to (4-6) with \( n = 2 \) ( - - - - - ) and to (4-7) ( - - - ).
Figure 4.3. Vertical profiles of horizontal velocities in a wind-water flume.
(a) to (h) corresponding to case No. 1 to 8 in table 4.1.
○ measurement (Yu, 1987); — DUTRID with equation (4-7).
Upon examining equation (4-6), it is easy to see that $|u^*| < |u^*|$ when $|u^*| > |u^*|$ and $|u^*| < |u^*|$ when $|u^*| > |u^*|$. Therefore, this equation yields a somewhat lower value of $\nu$ than equation (4-7) for finite $n$. However, it approaches equation (4-7) for $n \to \infty$.

The computational results using equation (4-7) are shown in figures 4.2 and 4.3. For all eight cases listed in table 4.1, the computed velocity distributions agree remarkably well with the measurements (see figure 4.3). Compared with the computations using equation (4-6) with $n = 2$, the agreement is equally good (see figure 4.2).

Summarizing, it was shown above that equations (4-6) with $n \geq 2$ and (4-7) yield the best fit of computed velocity profiles to the measurements. Since expression (4-7) is simpler, it is used in DUTRID as the default formulation.

### 4.4 Computations using the k-ε model

In order to examine the validity of the algebraic eddy viscosity model obtained in section 4.3 (equations (4-2) and (4-7)), an existing one-dimensional k-ε model for surface-gradient driven flows (Van Gent, 1991) was adapted and used to simulate the experiments of Yu (1987). The vertical profiles of the velocity computed with this model were compared to those computed with DUTRID as well as to the measurements. The computed vertical eddy viscosities were compared with equations (4-2) and (4-7).

The following equations are solved in the k-ε model to obtain the vertical profiles of flow velocities and the eddy viscosity.

1. Equation of momentum:

   $$\frac{\partial u}{\partial t} - \frac{\partial}{\partial z} \left( \nu \frac{\partial u}{\partial z} \right) = -g \frac{\partial H_z}{\partial x} ; \quad (4-8)$$

2. Equations for turbulent kinematic energy $k$ and dissipation rate $\epsilon$:

   $$\frac{\partial k}{\partial t} = c_1 \frac{k^2}{\epsilon} \left( \frac{\partial u}{\partial z} \right)^2 + \frac{c_1}{\sigma_k} \frac{\partial}{\partial z} \left( \frac{k^2}{\epsilon} \frac{\partial k}{\partial z} \right) - \epsilon \quad (4-9)$$

   $$\frac{\partial \epsilon}{\partial t} = c_{\mu \epsilon} k \left( \frac{\partial u}{\partial z} \right)^2 + \frac{c_1}{\sigma_\epsilon} \frac{\partial}{\partial z} \left( \frac{k^2}{\epsilon} \frac{\partial \epsilon}{\partial z} \right) - c_{\mu \epsilon} \frac{\epsilon^2}{k} \quad (4-10)$$
in which $c_l$, $c_{pc}$, $c_{ds}$, $\sigma_s$ and $\sigma$, are constants. The values of these constants were taken from Launder and Spalding (1972). $k$ and $\epsilon$ are related to the vertical eddy viscosity by $\nu = c_1 k^2 / \epsilon$.

The boundary conditions used are as follows.

(1) at the bottom boundary:

$u$: a logarithmic velocity distribution is assumed between $z = z_0$ and the adjacent grid point (law of the wall);

$k$, $\epsilon$: the turbulence is assumed to be in local equilibrium, or

$$c_1 \frac{k}{\epsilon} \left( \frac{\partial u}{\partial z} \right)^2 - \epsilon = 0$$

$$c_{pc} k \left( \frac{\partial u}{\partial z} \right)^2 - c_{ds} \frac{\epsilon^2}{k} = 0$$

(2) at the free surface boundary:

$u$: the turbulent shear stress is equal to the wind shear stress, or

$$|W| = \left[ \rho \nu \frac{\partial u}{\partial z} \right]_{z = H}$$

$k$, $\epsilon$: the same as for the bottom boundary.

A double-logarithmic grid is used in the $k$-$\epsilon$ model, with 60 levels used in all simulations. Computation begins from still water with suddenly imposed wind shear stress, and terminates when a steady state is reached.

Some of the computational results are shown in figures 4.4 and 4.5. It can be seen in figure 4.4 that the computed velocity distributions using the $k$-$\epsilon$ model compare quite well with the measurements.

The velocity profiles computed using DUTRID in which the simple algebraic eddy viscosity model given by equations (4-2) and (4-7) is applied, are also shown in the same figure. It is seen that they compare nearly as well with the measurements as the $k$-$\epsilon$ model results. This is seemingly in contradiction with significant differences in the eddy viscosity coefficients obtained from both models, shown in figure 4.5. For the case of following wind (figure 4.5c and d), the two models yield similar parabolic profiles with a difference in
Figure 4.4. Vertical profiles of horizontal velocities in a wind-water flume.
(a) to (d) corresponding to case No. 2, 4, 5, 6 in table 4.1.
○ measurement (Yu, 1987); - - - DUTRID with equation (4-7); — k-ε model.
Figure 4.5. Vertical profiles of vertical eddy viscosity in a wind-water flume.
(a) to (d) corresponding to case No. 2, 4, 5, 6 in table 4.1.
— - - - DUTRID with equation (4-7); — — k-ε model.
magnitude of about 25%. In view of the comparison of the corresponding velocity profiles to the measurements, it is difficult to judge which is better. For the case of opposing wind, the k-$\epsilon$ model results differ considerably from a parabolic distribution. Besides, the location of maximum viscosity is shifted from mid-depth towards the boundary with the larger shear stress (that is, the free surface in figure 4.5a and the bottom in figure 4.5b).

In order to explain the apparent discrepancy between the good agreement in the calculated velocity profiles and the moderate agreement in the calculated eddy viscosities, it is pointed out that in all cases the simple parabolic distributions of $\nu$ approach the k-$\epsilon$ model results quite well near the two boundaries where the velocity gradients are large. Large differences are found only in the central part of the water column where the velocity gradients are small. Therefore, the differences in the eddy viscosities given by both models have a quite small influence upon the velocity distribution, as is shown by the computed velocity profiles.

4.5 Conclusions and discussion

The flow model DUTRID developed in the present study was applied to compute the vertical distribution of horizontal velocities measured by Yu (1987) in a wind-water flume. The aim was to find a simple formulation of the vertical eddy viscosity suitable for flows driven mainly by wind and free surface gradient. By trying various expressions for the eddy viscosity, it was found that the conventional parabolic distribution over the depth gives a remarkably good fit to the measured velocity profiles:

$$\nu = \kappa |u^*| (z - H_b) \left[ 1 - \frac{z - H_b}{D} \right], \quad (4-2)$$

where the friction velocity $u^*$ can be equated to the larger of the bed- and surface-friction velocities, or

$$|u^*| = \max \left( |u_b^*|, |u_r^*| \right). \quad (4-7)$$

Computations also confirmed that an additive contribution from the free surface and bottom boundaries for $|u^*|$ results in an overestimation of the eddy viscosity. Equation (4-6)
with \( n \geq 2 \), on the other hand, gives good results.

Computations using a one-dimensional \( k-\epsilon \) model were made to verify the algebraic eddy viscosity model given by equations (4-2) and (4-7). When this viscosity model is applied, results of the velocity profiles obtained by using DUTRID compare equally well with the measurements as the \( k-\epsilon \) model results.

Wind waves were suppressed in the experiment of Yu by means of a surfactant (Teepol, SHELL detergent). The effects of wind waves upon turbulence are therefore not taken into account in the algebraic eddy viscosity model based on the experiment. In natural waters, however, wind waves are always present. Strong wind and storms generate intense turbulence in the fluid layer near the free surface boundary. Unfortunately, little is known about this kind of turbulence. In cases where waves are important, modifications of the presented formulation may be necessary. Such modifications will be most important for better prediction of the surface velocity.

The wind speeds and water velocities in Yu’s experiments are comparable to those found in practice, but the water depth is rather small. The Reynolds number is therefore moderate \( (10^4 \sim 10^5) \), and the flow was in the hydraulically smooth regime. In view of this the vertical eddy viscosity given by equations (4-2) and (4-7) should be applied in practice with due precautions.
CHAPTER 5

TEST COMPUTATIONS WITH DUTRID

5.1 Introduction

Numerical circulation models have to be tested and calibrated by using laboratory or field measurements before being applied to solve practical problems. It is useful to have at one's disposal detailed measurements of flow variables, such as the velocity and free-surface set-up for some typical flow cases. Such benchmark measurements could be used as "standard tests" for any numerical circulation model. However, there are few measurements available in the literature for this purpose. One of the main reasons is that experiments are costly and demand considerable efforts. It is particularly difficult to make detailed simultaneous measurements of velocity distributions in an unsteady flow at a large number of points in the flow field. Besides, field measurement of flows under extreme weather conditions, which are the most interesting cases for verification of a flow model, is a challenging work. Because of the absence of the above-mentioned benchmark experiments, numerical circulation models are often tested only for some typical flows under rather idealized conditions.

In this chapter the flow model DUTRID developed in the present study is applied to simulate some flows under simplified conditions. One of the aims was to examine the numerical accuracy and computational efficiency of the flow model. The four flow cases presented in the next section concern uni-directional flows in straight channels. Existing analytical solutions of velocity distributions for some of these flows are compared with computations. In section 5.3 secondary currents caused by the Coriolis force in horizontal flows are simulated and compared with analytical solutions given by Kalkwijk and Booij (1986). Section 5.4 deals with the numerical simulation of velocity distributions in a model
harbour basin. The results are compared with laboratory measurements made by Langendoen and Kranenburg (1990). Some conclusions are presented in section 5.5.

5.2 Uni-directional flows

5.2.1 Pressure-gradient driven flow

For pressure-gradient driven flow we consider a steady flow in a straight channel, infinitely wide and with a constant small bottom slope, see figure 5.1. Wind shear stress at the free surface is assumed equal to zero. The Coriolis force in the model equations is neglected. Assuming a vertical distribution of the eddy viscosity \( \nu \), a steady-state analytical solution can be found for the vertical distribution of the velocity.

Under these conditions and assuming a uniform flow, equation (3-21) used in the velocity profile module can be simplified as follows.

\[
\frac{\partial}{\partial z} \left( \nu \frac{\partial u}{\partial z} \right) = g \frac{\partial H_s}{\partial x} .
\]  

(5-1)

The boundary conditions are as follows.

* At the free surface \( (z = H_s) \): \( \tau_z = 0 \), or \( \nu \frac{\partial u}{\partial z} = 0 \);
* At the bottom \( (z = H_s + z_0) \): \( u = 0 \) (no-slip condition).

A parabolic vertical distribution can be assumed for the eddy viscosity (see chapter 4):

\[
\nu = \kappa u^* (z - H_s) \left( 1 - \frac{z - H_s}{D} \right) ,
\]  

(5-2)

where the velocity scale of turbulence \( u^* \) can be estimated by using the bottom shear stress:

\[
u^* = \left( \frac{|T|}{\rho} \right)^{1/2} .
\]

Since the wind shear stress is equal to zero and a steady-state flow is assumed, the forces in the flow direction acting on a water column are at equilibrium, or \( |T| = \rho g D |\partial H_s/\partial x| \).

As a result, \( u^* = \sqrt{g D |\partial H_s/\partial x|} \).
Figure 5.1. Uniform pressure-gradient driven flow in a straight channel.

Commutations with 5 levels (- - - - -), 10 levels (--- ---), and 20
levels (----- -----) in the vertical; analytical solution (-----).

By using the boundary conditions and the expression of $u^*$ given above, equation (5-1)
can be easily integrated twice with respect to $z$ to obtain the following velocity distribution:

$$u = \frac{u^*}{\kappa} \ln \left( \frac{z - H_b}{z_0} \right) \quad (5-3)$$

The following values were assumed in the test computations with DUTRID: $g = 9.81$
m/s$^2$, $D = 5.0$ m, $\delta H/\delta x = 0.0001$, $z_0 = 0.01$ m. A grid of 5×3 was used in $x$- and $y$-
directions. The grid spacing was 500 m. A time-step of 100 s was used. As boundary
conditions the free surface levels at both ends of the channel were kept constant.
Computations commenced from still water conditions with a free surface parallel to the
bottom, and continued until a steady state was reached.

Numerical results of velocity profiles at the centre of the channel are shown in figure
5.1, with 5, 10 and 20 vertical levels used in the computations, respectively. Linear
interpolation was used between grid points, except between the lowest two points where a
logarithmic velocity profile was assumed. This convention is used in other plots of velocity
profiles presented in this chapter as well. The analytical solution given by equation (5-3) is
also plotted in the figure. It is seen that the computational results are quite accurate, even for a very small number of grid points in the vertical. The large velocity gradients near the bottom are well reproduced.

One may conclude from this test case that the approximations of the bottom boundary condition made in DUTRID by using the Law of the Wall, are quite adequate.

5.2.2 Surface shear induced flow

For surface shear induced flows we consider a flow in a straight channel, infinitely wide and with a horizontal bottom, see figure 5.2. A constant uniform wind shear stress \( \tau \) is suddenly imposed on the free surface in the longitudinal direction. A net discharge in the wind direction is allowed, such that the free surface gradient is equal to zero. The Coriolis force is assumed equal to zero. For a given vertical distribution of the eddy viscosity \( \nu \), a steady-state analytical solution can be found for the vertical distribution of velocity.

Under these conditions the flow equation simplifies to

\[ \frac{z}{D} \]

\[ u \] (cm/s)

\[ 0 \]

\[ 0.0 \]

\[ 0.2 \]

\[ 0.4 \]

\[ 0.6 \]

\[ 0.8 \]

\[ 1.0 \]

\[ 0 \]

\[ 10 \]

\[ 20 \]

\[ 30 \]

Figure 5.2. Wind-driven flow in a straight channel. For legend, see figure 5.1.
\[
\frac{\partial}{\partial z} \left( \nu \frac{\partial u}{\partial z} \right) = 0 .
\] (5-4)

The boundary conditions are as follows.

* At the free surface \( z = H_s \): \( \tau_s = |W| \), or \( \nu \frac{\partial u}{\partial z} = \frac{|W|}{\rho} \);

* At the bottom \( z = H_b + z_0 \): \( u = 0 \) (no-slip condition).

A parabolic vertical distribution for the eddy viscosity \( \nu \) can be assumed (equation (5-2)). The velocity scale of turbulence in (5-2) can be estimated by using the wind shear stress:

\[
u^* = \left( \frac{|W|}{\rho} \right)^{1/2}.
\]

For the boundary conditions and the expression of \( u^* \) given above, the solution for \( u \) in equation (5-4) is

\[
u = \frac{u^*}{\kappa} \ln \left( \frac{z - H_b}{D - (z - H_b)} \frac{D - z_0}{z_0} \right) .
\] (5-5)

Note that in this solution a singularity exits at \( z = H_s = D + H_b \) where \( u \to \infty \). The singularity is associated with the assumed eddy viscosity approaching zero near the free surface.

The following values were assumed in the test computations: \( g = 9.81 \text{ m/s}^2 \), \( D = 5.0 \text{ m} \), \( |W| = 0.173 \text{ N/m}^2 \) (corresponding to a wind speed of about 10 m/s), \( z_0 = 0.01 \text{ m} \). A grid of \( 5 \times 3 \) was used in the longitudinal and transverse directions, respectively. The grid spacing was \( 500 \text{ m} \). A time-step of 100 seconds was used in the 2DH module. It was possible to use a larger time-step in the velocity profile module as is typical with wind-driven flows (see section 3.6.1). In this simulation the used time-step was 100 s till \( t = 500 \text{ s} \), 200 s till \( t = 10000 \text{ s} \) and 500 s till \( t = 50000 \text{ s} \). As boundary conditions, the free surface levels at both ends of the channel were kept constant and equal to their values at still water. Computations commenced from still water conditions and continued until \( t = 50000 \text{ s} \), when a steady-state was nearly reached.

Numerical results of velocity distributions at the centre of the channel are shown in figure 5.2, with 5, 10 and 20 grid points in the vertical, respectively. The analytical solution given by equation (5-5) is also shown in this figure. It is seen that the numerical results approximate the analytical solution with excellent accuracy, even with 5 levels in the vertical.
The large velocity gradients near the bottom, and near the free surface where the singularity exists, are well reproduced. Note that the grid point nearest to the free surface is half a grid-spacing below (see figure (3-2)), so that the theoretically infinite velocity at the free surface does not hamper the computations.

The computations presented in this section demonstrate that the vertical discretization and the approximations of the free surface boundary condition used in the velocity profile module (see sections 3.5.2 and 3.5.3), are quite adequate. The computations also show that a larger time-step can be used in the velocity profile module than that used in the depth-averaged flow module, at least for flows characterized by the slow process of momentum transfer from wind shear down the water column.

### 5.2.3 Wind-driven flow in a closed channel

The flow considered in this section is similar to that simulated in the preceding section. In this case, however, the channel is closed with vertical walls at both ends so that the net flow rate is equal to zero when a steady-state is reached. Far from the end-walls flows in the wind direction are generated in the fluid layers near the free surface through wind action. A return current is present in the lower part of the water column. A circulating flow in the vertical plane then results, see figure 5.3.

Under the above conditions the steady-state model equation (5-1) holds except in regions of the channel near the end-walls. This equation is subjected to the boundary conditions as follows.

* At the free surface \((z = H_f)\): \(\tau_x = |W|, \) or \(\nu \frac{\partial u}{\partial z} = \frac{|W|}{\rho} \);
* At the bottom \((z = H_b + z_0)\): \(u = 0\) (no-slip condition).

Assuming a parabolic vertical distribution of the eddy viscosity \(\nu\) (equation (5-2)), the following solution can then be obtained:

\[
\begin{align*}
  u &= \frac{1}{\kappa u^*} \left[ \frac{|W|}{\rho} \ln \left( \frac{z - H_b}{D - z + H_b} \frac{D - z_0}{z_0} \right) - gD \frac{\partial H_x}{\partial x} \ln \left( \frac{z - H_b}{z_0} \right) \right], \\
  u^* &= \sqrt{\frac{|W|}{\rho}}.
\end{align*}
\]
Figure 5.3. Wind-driven flow in a closed channel. (a) $t = 200$ s; (b) $t = 400$ s; (c) $t = 800$ s; (d) $t = 2000$ s; (e) steady-state analytical solution.

Since the channel is closed at both ends, the constraint $Q_x = 0$ applies for all sections when a steady-state is established. The free surface gradient can be determined using this condition. Integrating equation (5-6) over the water depth and using $Q_x = 0$, one obtains:

$$Q_x = \frac{1}{\kappa u^*} \left[ \frac{|W|}{\rho} D \ln \left( \frac{D}{z_0} \right) - g D^2 \frac{\partial H_z}{\partial x} \left( \ln \left( \frac{D}{z_0} \right) - 1 \right) \right] = 0.$$

The following values were assumed in the test computations: $g = 9.81$ m/s$^2$, $D = 5.0$ m, $|W| = 0.173$ N/m$^2$, $z_0 = 0.01$ m. At the up- and downstream boundaries a zero flow rate was imposed. A grid of $11 \times 2$ was used in the horizontal directions, with 20 grid points in the vertical. The horizontal grid spacing was 50 m. A time-step of 50 second was used. Computations began from still water conditions. After several periods of long wave oscillations in the channel the flow stabilized and the steady-state was gradually reached.

Numerical results of velocity profiles at the centre of the channel are shown in figure 5.3, corresponding to time $t = 200$ s, 400 s, 800 s and 2000 s after the wind is imposed on the free surface, respectively. The steady-state analytical solution given by equation (5-6) is also shown in the figure (curve e). It is seen that the computed velocity profiles gradually converge to the steady-state analytical solution. The large velocity gradients near the bottom
and the free surface, where a singularity exists, are well reproduced.

Upon examining the computed velocity distributions near the end-walls of the channel, it was found that they are almost identical to those in the central part of the channel. The influence of the end-walls upon flows is not clearly seen. However, it is well known that such influence should be limited to a distance from the end-walls approximately equal to the local water depth. In the test computations, the horizontal grid spacing was 50 m and the water depth was only 5 m. Since the nearest grid points to the end-walls are at a distance 5 times the water depth away, it is reasonable that the influence of the end-walls is not noticeable in the computed velocity distributions at those grid points.

It should be noted that the bottom shear stress formulation used in a typical 2DH flow model is based on the depth-averaged flow rates. The defect of such a formulation is most clearly illustrated by the present example. Following this formulation the bottom shear stress would be equal to zero. As a consequence of the error in estimating the bottom shear stresses, the free surface gradient would be overestimated by about 30% depending on the assumed bottom roughness.

5.2.4 Flow over a bed of nonuniform slope

Besides the external driving forces such as wind shear and a pressure gradient, the bottom topography also affects the velocity distribution in a flow field. In the preceding sections, flows over a bottom with a zero or constant bottom slope were examined. In this section velocity distributions in a free-surface gradient driven flow in a section of a channel with a gradual deepening in the flow direction (see figure 5.4) are considered.

As was shown in section 3.3.3, the advective terms in the model equations are usually small for shallow water flows with low Froude number. However, as was demonstrated through a two-dimensional vorticity analysis (section 3.4.2), they nevertheless may have marked effects upon flows. For the flow considered in this section, the bottom slope is small so that these terms are not likely to be large. It is useful to see how they affect the velocity
distributions. Therefore, computations were made with advective terms being either included or neglected.

In the sloping part of the channel the bed slope varies in the flow direction following a cosine function. The maximum bottom slope is 1.885%. As boundary conditions a flow rate of 0.8 m$^3$/s was imposed at both ends of the channel. The bottom roughness ($z_0$) was assumed equal to 0.005 m. The flow field is discretized into a grid of $53 \times 2 \times 20$ in the longitudinal, transverse and vertical directions, respectively, with the horizontal grid spacing equal to 25 m. The time-step used was 10 s in the depth-averaged module, and 20 s in the velocity profile module. Computations begin from still water conditions. The steady-state is gradually reached after several periods of long wave oscillations.

The computed steady-state results for the velocity profiles at five different cross-sections in the channel are shown in figure 5.5. The locations of these sections are marked in figure 5.4. Both results of computations with the convective terms being either neglected or taken into account are shown for comparison.

As can be expected, typical logarithmic velocity distributions are found at sections of the channel upstream and far downstream of the sloping part. At these sections the velocity distributions are nearly uniform in the longitudinal direction so that the advective terms approach zero.

Marked differences are found between the two cases at other sections of the channel. With the advective terms being neglected, the computed velocity profiles at all sections of
Figure 5.5. Velocity distributions of flow over a sloping bed.

- - - advective terms included;
--- advective terms neglected.
the channel look similar. The magnitude of the velocity depends solely on the local water depth. With the advective terms being taken into account, the velocity profiles at sections in the deepening part are 'smoothed-out' in the flow direction in the sense that velocities vary in a more gradual way in that direction. This is a natural consequence of inertia. Typical nearly linear distributions at some distance above the bottom can be seen at these sections.

When the advective terms are neglected in the flow equations ((3-21) and (3-22)) used to solve the vertical distributions of velocities, the computations at all points in the horizontal grid are independent of one another. With the local free-surface gradients obtained in the depth-averaged flow module, the vertical distribution of velocities can be locally determined. Therefore, similar logarithmic distributions are obtained at all sections of the channel. With the advective terms being taken into account, computations of the velocities at all cross-sections are related to each other. Fluid parcels from regions of high momentum conserve part of this momentum when entering regions of low momentum, which reduces the spacial differences in velocities in the flow direction ('smoothing').

The bottom slope assumed in this example is moderate. However, the computational results show that the influence of advective terms upon the flow is still noticeable. Since bottom slopes of this magnitude are often found in practice, the advective terms should be retained in the model equation, which conforms to the analysis of the relative importance of these terms in the basic equations presented in section 3.4.2. In general, it should be noted that since shallow water is assumed in the present flow model, flows over a bed with large bottom slope in which the vertical accelerations are large, cannot be adequately simulated. An artificial 'smoothing' of the bottom may be necessary at those locations. It is expected that the effects of such smoothing upon the flow are confined to the vicinity of such locations so that the overall flow pattern is little affected.

5.3 Secondary current in nearly horizontal flow

In natural waters, both curvature of the main flow and the Coriolis acceleration give rise to deviations in the directions of local flow velocities with respect to the direction of the
depth-averaged one. The horizontal velocity vector typically rotates over the vertical, which phenomenon is often referred to as secondary current (Kalkwijk and De Vriend, 1980). Although in most cases the magnitude of the velocities involved is quite small compared to those of the main flow, the secondary current may have significant influence upon such processes as the transport of sediments near the bed and the dispersion of solutes.

In this section the flow model DUTRID is used to simulate the secondary current caused by the Coriolis force in nearly horizontal flow. Secondary currents caused by curvature of the main flow or by Coriolis acceleration are virtually identical (Kalkwijk and Booij, 1986), and both can be simulated using the present model. However, when simulating secondary current caused by flow curvature, such as in flow in a river bend, a large number of finite-difference grid points may be needed to represent the curvature with sufficient accuracy. Therefore, secondary current in a section of a straight channel caused by the Coriolis force (see figure 5.6) was simulated instead.

![Figure 5.6. Secondary current in nearly horizontal flow.](image)

In Kalkwijk and Booij (1986) the following assumptions were made for the flow under consideration:

1) the vertical distribution of the main flow velocity is logarithmic and is not influenced by the secondary current;
2) the vertical eddy viscosity is completely determined by the main flow velocities;
3) the flow is locally adapted.

Under these assumptions, the solution for the transverse velocity was found:

\[
v = -\text{sign}(Q_a) \frac{C_o D}{k^2} f_c \left[ \frac{z - H_b}{D}, \alpha \right], \tag{5-7}
\]
Figure 5.7. Secondary current caused by the Coriolis force,
(1) analytical solutions (Kalkwijk and Booij, 1986); (2) computations using DUTRID.
in which \( \alpha \) is a friction parameter given by

\[
\alpha = \frac{\sqrt{g}}{\kappa C} = \left[ \ln \frac{D}{z_0} - 1 \right]^{-1},
\]

in which \( C \) = coefficient of Chézy (in \( m^{1/2}/s \)) and \( f_c \) is a function shown in figure 5.7a.

A straight channel of 200.0 m \( \times \) 50.0 m with a flat bottom was chosen for the test computations. The initial water depth was 5.0 m. A flow rate of 2.5 m\(^2\)/s was imposed at both ends of the channel. The Coriolis coefficient was equal to \( 10^{-4} \) s\(^{-1}\). The coefficient of Chézy ranged from 30 to 70 m\(^{1/2}\)/s. The flow field was discretized into a grid of \( 20 \times 5 \times 40 \) in the longitudinal, transverse and vertical directions, respectively. Steady solutions were obtained after several periods of long wave oscillations. Far from the ends and sidewalls of the channel the secondary flow velocities become uniform. The results of the transverse velocity \( v \) were scaled with a factor \(-C_v D/\kappa^2\) and plotted in figure 5.7b. This scaled velocity is equivalent to the function \( f_c \) in equation (5-7) plotted in figure 5.7a.

It is seen in figure 5.7 that the agreement between the analytical and numerical results is quite good. The vertical positions of zero secondary current and the sharp velocity gradients near the bottom are well reproduced by the present flow model. Nevertheless, the sharp velocity gradient gave rise to a difference in the transverse velocity of about 5% (\( C = 30 \) m\(^{1/2}\)/s) to 15% (\( C = 70 \) m\(^{1/2}\)/s) for the chosen number (40) of grid points in the vertical. Near the free surface, the difference is less than 5%.

### 5.4 Flow in a model tidal harbour basin

In this section the flow model DUTRID is applied to simulate circulations in a model harbour basin opening to a model river in which a uniform oscillatory flow is present, see figure 5.8. The model geometry is identical to one of the models examined by Langendoen and Kranenburg (1990) in laboratory observations of the velocity field in the harbour and the exchange of matter between harbour and river.

In the experiment the harbour basin was built alongside of a flume, 18 m long and 1 m wide. The bottom of the model is horizontal and the side walls are vertical. Oscillatory
currents were created in the flume using a constant water supply and adjustable weirs at each end of the flume. Measurements of local velocities were made by using an electromagnetic flow meter. The reader is referred to Langendoen and Kranenburg (1990) for further details of the experiments. The current was without water level changes for the experiment simulated here.

![Diagram](image-url)

**Figure 5.8. Flow in a model harbour basin.**

Since the emphasis of this numerical study was on the velocity distributions inside the harbour, a steady-state flow was considered in the computations. A flow rate linearly increasing from zero at time \( t = 0 \text{ s} \) to \( 0.041 \text{ m}^3/\text{s} \) at \( t = 200 \text{ s} \) and then remaining constant, was assumed at both ends of the flume. A grid of \( 21 \times 40 \) was used to represent the model harbour and river, see figure 5.9. The horizontal grid spacing was 0.1 m. Five vertical levels were used.

In the computations a time-step of 0.05 s was used. This corresponds to a Courant number equal to about 0.5. The Courant number is defined as

\[
\frac{c \Delta t}{\Delta x}
\]

in which \( \Delta x = \) horizontal grid-spacing; \( c = \) propagation speed of long waves \( (c = \sqrt{gD}) \).

Computations using a much larger time-step showed numerical instability starting from the mixing zone between the harbour and the river. This could be attributed to the fact that the horizontal exchange terms in the basic equations, which are retained in the 2DH module but neglected in the velocity profile module, are locally quite large near the mixing zone.
Figure 5.9. Finite-difference grid used for modelling circulations in a model harbour basin. The area enclosed with dashed lines is used to visualize the flow.
Computations showed that the circulations in the harbour basin depend markedly on the horizontal eddy viscosity $E$. This is not surprising since the circulation in the basin is entirely driven through horizontal momentum transfer across the mixing zone. For the results shown below, a constant value \(5.0 \times 10^{-4} \text{ m}^2/\text{s}\) is assumed for $E$. This value results from computations of steady-state depth-averaged flows in the model harbour in which turbulence was modelled by using a $k$-$\varepsilon$ model (R. Booij, 1989).

A constant value \((0.05 \text{ mm})\) was assumed for the bottom roughness height $z_0$. From measurements in the physical model of the free surface slope in the river under steady-flow conditions, a coefficient of Chézy equal to 51.1 \(\text{ m}^{1/2}/\text{s}\) was obtained (Langendoen and Kranenburg, 1990). The corresponding roughness height is slightly larger than that assumed here.

In figure 5.10 the vector plots of computed steady-state horizontal velocities at five different vertical levels are shown for an area around the harbour which is marked in figure 5.9. The advective terms were included in the model equations. The flow patterns in the harbour look quite complex. As far as the main flows are concerned, the velocities generally increase with elevation with maximum currents at the free surface. Two distinctive features of the circulation in the harbour may be observed upon examining the results in greater detail. Firstly, secondary currents caused by curvature of the main flow around the corners of the sidewalls can be clearly seen. One finds a net volume flux towards the centre of the basin in the bottom layers and a flux leaving the centre in the upper layers near the free surface.

The second feature in the computed circulations in the basin is quite remarkable. In the area of the basin where the incoming river water meets the downstream sidewall (near the stagnation point), the flow is strongest near the bottom.

The above steady-state solution of the velocity distribution in the harbour can be compared with observations of Langendoen and Kranenburg with a sinusoidal current in the flume, at maximum current. The amplitude of the flow rate was equal to 0.041 \(\text{ m}^3/\text{s}\), the same as in the numerical simulations. The period was 500 s. The measurements of velocities at four different vertical elevations for the same area as was marked in figure 5.9, are shown
Figure 5.10. Computed circulations in a model harbour basin, advective terms included. (a) 2 cm above the bottom; (b) 4 cm above the bottom; (c) 6 cm above the bottom; (d) 8 cm above the bottom; (e) 10 cm above the bottom.
Figure 5.11. Circulations in a model harbour basin. Measurements by Langendoen and Kranenburg (1990). (a) 1.5 cm above the bottom; (b) 4 cm above the bottom; (c) 6 cm above the bottom; (d) 8 cm above the bottom.
in figure 5.11. The general agreement between the computations and the measurements is good. The two above-mentioned features of the circulations in the harbour are also found in the observations.

The physical mechanism underlying the second feature of the circulation is still not fully understood. The phenomenon is possibly caused by the inflow of higher momentum fluid near the bottom from the mixing layer between the river and the harbour. The circulating flow in the harbour has considerably lower momentum than the incoming river flow. The secondary current near the bottom, which is directed towards the centre of the harbour, might guide the high momentum fluid into the harbour.

In order to examine the effect of advective terms, for this particular case, computations were also made with these terms being set equal to zero. The results are shown in figure 5.12. The general pattern of circulating flow in the harbour is well simulated. However, the two secondary current features mentioned above are not seen. One may hence conclude that those important features of circulations in the harbour are determined by the advective transport.

Discussion

(1) The depth-to-width ratio of the harbour basin simulated is approximately 9. The shallow water assumption is therefore approximately satisfied. The computations and measurements presented above show that advective terms are nevertheless quite important for the flows considered. These terms are responsible for the secondary current in the harbour basin. As a possible consequence in practice, this current could imply an accumulation of sediments like silt in the central part of the basin.

(2) Analyzing the computational results one can find that the secondary current and other deviations of the velocity from the main circulating flow velocities in the basin are strongest near the sidewalls. The velocities decrease rapidly from the sidewalls towards the centre of the basin. This is in accordance with the general assertion that effects of sidewalls upon velocity shear should be confined to the vicinity of the walls. On the other hand, the
Figure 5.12. Computed circulations in a model harbour basin, advective terms neglected.
For legends, see figure 5.10.
horizontal momentum exchange terms neglected in the velocity profile model of DUTRID, are likely to be important near the sidewalls because of the large horizontal gradients of velocities. These terms may locally affect the accuracy of the computed vertical distributions of velocities.

5.5 Conclusions

In this chapter the numerical flow model DUTRID was applied to simulate flows with simple geometries. The following conclusions can be drawn:

(1) For uni-directional flows in straight channels induced by a free-surface gradient, by surface wind shear or by a combination of both, analytical solutions exist for the velocity distributions assuming a vertical distribution of the eddy viscosity \( \nu \). The velocity distributions of these flows computed with DUTRID approximate the analytical solutions with good accuracy, even for a small number of vertical levels. Large velocity gradients near the boundaries are well reproduced by the flow model.

(2) For circulations in shallow water with a low Froude number, the advective terms in the equations of motion are usually small. However, they are likely to have noticeable consequences upon flow characteristics in certain cases. The computation of the flow in a channel with a maximum bottom slope of about 2\% showed that the vertical distributions of velocities are markedly influenced by these terms.

(3) Secondary currents induced by the Coriolis force in a straight channel with a horizontal bottom were computed using the model. These currents have the same characteristics as those induced by curvature of the main flow. The results agree well with analytical solutions.

(4) Computations of circulations in a shallow laboratory harbour basin opening to a tidal river also show that the advective terms have marked effects upon the flow characteristics. They are responsible for some distinct flow features of the flow in the basin,
in particular the secondary flows. These features were also observed in the laboratory experiment by Langendoen and Kranenburg (1990).
CHAPTER 6

A PARTICLE MODEL FOR DISPERSION OF SOLUTES

6.1 Introduction

In natural water bodies flows are always present because of the action of wind, tidal forces, inflows and outflows. These flows convey and diffuse nutrients and sediment particles, waste discharges from human activities such as sewage water, trace metals and chemicals. These constituents are mixed with the environmental water through the combined effects of shear and transverse turbulent diffusion. This process is usually called dispersion. It largely determines the fate of the constituents and hence the water quality. Mathematical modelling of transport and diffusion of substances is of great interest for engineering applications.

As part of the present study concerning numerical modelling of flow and dispersion in shallow water, a dispersion model of solutes and fine sediments has been developed. The model runs in conjunction with the flow model DUTRID. Only passive dissolved substances were considered, with the aim to concentrate on the two main dispersion processes, namely, advection and turbulent diffusion. For this purpose the substances were considered as marked water parcels which have no effect on the water flow. As possible extensions of the model, other aspects of dispersion, such as the decay of substances, can easily be incorporated. A fall velocity of particles can also be included for modelling deposition of sediments.

The advection-diffusion equation is most commonly used to describe the dispersion of solutes in turbulent flows. As was shown by Taylor (1921), this description is valid provided that the size of the solute cloud is much larger than the Lagrangian integral length-scale of
the turbulence. For turbulent diffusion in shallow water flows, this length-scale is of the order of the water depth, and hence is small with respect to the size of the solute cloud. Therefore, the advection-diffusion equation can be applied. When large turbulence eddies in the horizontal plane are present in the flow, however, the size of the solute cloud may be relatively small and the application of the advection-diffusion equation is less appropriate. Such turbulence may be generated near a lateral mixing layer.

The usual approach to model dispersion has been to solve the advection-diffusion equation on a Eulerian grid by using a Finite-Difference Method or a Finite-Element Method. However, the particle tracking method is used in this study. Particle models have a number of significant advantages compared to models based on the usual approach (see section 2.2). In particular, no numerical diffusion is introduced and no negative concentrations are produced in a particle model. These properties are particularly relevant when modelling the near-field dispersion of a local discharge, accidental for example, where the concentration gradients are large.

Details of the particle model adopted are presented in this chapter. In the next section the theoretical aspects of the particle tracking method are briefly described, followed by an outline of the model. Subsequently, modelling of diffusion in homogeneous and nonhomogeneous turbulence is presented in section 6.3 and illustrated by means of a computational example. A cell-analytical approach to model advection numerically is presented in section 6.4. It is compared with a classical numerical integration algorithm for pure advection of particles in a rigidly rotating flow field (Molenkamp test). Details of numerical simulations of two-dimensional shear dispersion for which an analytical solution exists, are presented in section 6.5. Possibilities to increase the computational efficiency of the particle model applied to dispersion in shallow waters are examined. Some conclusions are given in section 6.6.
6.2 The particle tracking model

6.2.1. Theoretical formulation

Theory

With the particle tracking method the total mass of solutes dispersed in a fluid flow is represented by a large number of discrete point masses, or particles, with equal amounts of matter. These particles are displaced in the flow field following prescribed deterministic and random movements that simulate advection and turbulent diffusion of solutes, respectively. The concentration distribution of solute at a given instant is approximated by the distribution of particle masses in the flow field. For the theoretical formulation we need to consider only one single particle. The distribution of total particle mass is then represented by the probability density \( P \) of the position of that particle.

In the following the basic equations of random walk are introduced. These are the Langevin equation describing the movement of the particle, and the Fokker-Planck equation describing the probability density \( P \). The Fokker-Planck equation is similar to the advection-diffusion equation. From this analogy equations are obtained that relate the displacements of the particle to the fluid velocity and diffusivity used in the advection-diffusion equation.

Using a vector \( \mathbf{X} = (X_i, \ i = 1,2,3) \) to represent the position of a particle in a rectangular coordinate system, the random movement of the particle is described by the Langevin equation (Langevin, 1908):

\[
\frac{d\mathbf{X}}{dt} = F_i(\mathbf{X}) + G_j(\mathbf{X}) \eta_j(t) , \ (i,j = 1, 2, 3)
\]  

(6-1)

in which \( F_i \) and \( G_j \) are prescribed functions. They are related to the advection velocity and turbulent eddy viscosity, respectively (see further below); \( \eta_j(t) \) is white noise (Papoulis, 1965). \( G_j \) represents the intensity of the white noise and is called the Langevin force. The overhead tilde \((\sim)\) denotes a random variable. The summation convention is used in this equation and in other equations throughout this section.

In general the velocity and diffusivity distributions in a fluid flow are nonuniform. \( F_i \) and \( G_j \) are therefore functions of \( \mathbf{X} \), in which case (6-1) becomes a non-linear stochastic
differential equation. Still, this equation is incomplete. A definition must be given as to whether $G_y$ should be determined before the random jump (Itô convention) or after (Stratonovich convention), in order for equation (6-1) to define a unique stochastic process (Van Kampen, 1981). The Itô convention is mostly used because it can be realized using an explicit algorithm (Uffink, 1990). It is also used in the following analysis.

Formal integration of equation (6-1) gives:

$$
\Delta \bar{X}_i = F_i(\bar{X}) \Delta t + G_y(\bar{X}) \bar{W}_i(\Delta t),
$$

(6-2)

where $\Delta t \rightarrow 0$; $\bar{W}_i(\Delta t)$ is the integrated white noise or Wiener process. It has zero mean and (co-)variance

$$
< \bar{W}_i(\Delta t) \bar{W}_j(\Delta t) > = \delta_{ij} \Delta t.
$$

(6-3)

where angular brackets "<..>" denote expectation value or mean.

In a random walk the particle undergoes successive steps of random displacements $\bar{S}_i$ in each finite time interval $\Delta t$. $\bar{S}_i$ is called the step function of the random walk. Supposing the particle is located at $X$ at time $t$, the step function is defined as (c.f. equation (6-2))

$$
\bar{S}_i(X) = F_i(X) \Delta t + G_y(X) \bar{W}_i(\Delta t),
$$

The mean and the mean cross-product of $\bar{S}_i(X)$ can easily be derived, using (6-3):

$$
< \bar{S}_i(X) > = F_i(X) \Delta t + G_y(X) < \bar{W}_i(\Delta t) > = F_i(X) \Delta t;
$$

$$
< \bar{S}_i(X) \bar{S}_j(X) > = G_{y_i}(X) G_{y_j}(X) \Delta t + F_i(X) F_j(X) \Delta t^2.
$$

(6-4)

The physical meanings of $F_i$ and $G_y$ become evident from the above equations: $F_i$ is the mean displacement of the particle per unit time. It obviously corresponds to the advective flow velocity. $G_y G_{jk}$ is the increase in the mean cross-product of the step function per unit time. Its correspondence to the turbulent diffusivity will become more evident further below.

The coordinate vector $\bar{X}(t)$ of the particle can be considered as a stochastic process. Defining a probability density ($P$) of $\bar{X}$ by:

$$
P = \lim_{\Delta V \rightarrow 0} \frac{1}{\Delta V} Pr \left\{ X - \frac{\Delta X}{2} \leq \bar{X}(t) \leq X + \frac{\Delta X}{2} \right\}, \quad \Delta V = \Delta X_1 \Delta X_2 \Delta X_3;
$$

it can be proved that $P$ satisfies the Fokker-Planck equation (Feller, 1971; Van Kampen, 1981) which can be written as follows:
\[
\frac{\partial P}{\partial t} = - \frac{\partial}{\partial X_i} (A_i P) + \frac{\partial^2}{\partial X_i \partial X_j} (B_{ij} P) ,
\]

where \(A_i\) is called the drift vector and \(B_{ij}\) the noise tensor. They are defined as:

\[
A_i(X) = \lim_{\Delta t \to 0} \frac{1}{\Delta t} < \ddot{S}_i(X) > ;
\]

\[
B_{ij}(X) = \lim_{\Delta t \to 0} \frac{1}{2 \Delta t} < \ddot{S}_i(X) \ddot{S}_j(X) > .
\]

\(A_i\) and \(B_{ij}\) are related to \(F_i\) and \(G_{ij}\) in equation (6-1). Substitution of (6-4) into (6-6) yields:

\[
A_i = F_i ;
\]

\[
B_{ij} = \frac{G_{ik} G_{jk}}{2} .
\]

The Fokker-Planck equation (6-5) is similar to the advection-diffusion equation which can be written as follows:

\[
\frac{\partial c}{\partial t} = - \frac{\partial}{\partial X_i} (v_i c) + \frac{\partial}{\partial X_i} \left( D_{ij} \frac{\partial c}{\partial X_j} \right) ,
\]

where \(c =\) concentration; \(v_i =\) flow velocity; \(D_{ij} =\) diffusivity tensor. Using the Reynolds analogy, \(D_{ij}\) is equal to the turbulent eddy viscosity.

Equation (6-5) can be rearranged as follows to allow for a direct comparison with the advection-diffusion equation:

\[
\frac{\partial P}{\partial t} = - \frac{\partial}{\partial X_i} \left( A_i - \frac{\partial B_{ij}}{\partial X_j} \right) P + \frac{\partial}{\partial X_i} \left( B_{ij} \frac{\partial P}{\partial X_j} \right) .
\]

Equivalency of (6-8) and (6-9) requires:

\[
P = c ; \quad B_{ij} = D_{ij} ; \quad A_i = v_i + \frac{\partial D_{ij}}{\partial X_j} .
\]

Substitution of equations (6-7) into the above relations yields:

\[
F_i = v_i + \frac{\partial D_{ij}}{\partial X_j} ;
\]

\[
\frac{G_{ik} G_{jk}}{2} = D_{ij} .
\]
For a flow field with given velocity \( v_i \) and diffusivity \( D_{ij} \), a random walk is uniquely defined by equations (6-1) and (6-10). The probability density of the random walk is then equivalent to the concentration of solute dispersed in that flow field.

It is convenient to choose coordinate axes which coincide with the principal axes of the diffusivity tensor \( D_{ij} \) (or the eddy viscosity). In that case, (6-10) becomes:

\[
\begin{align*}
F_x &= u + \frac{\partial D_{xx}}{\partial x} ; \\
F_y &= v + \frac{\partial D_{yy}}{\partial y} ; \\
F_z &= w + \frac{\partial D_{zz}}{\partial z} ;
\end{align*}
\]

\[
G_{xx} = \sqrt{2D_{xx}} ; \\
G_{yy} = \sqrt{2D_{yy}} ; \\
G_{zz} = \sqrt{2D_{zz}} ;
\]

(6-11)

and the step function (see equation (6-2)) is:

\[
\begin{align*}
S_x &= \Delta \tilde{X}_1 = F_x \Delta t + G_{xx} \tilde{W}_x (\Delta t) ; \\
S_y &= \Delta \tilde{X}_2 = F_y \Delta t + G_{yy} \tilde{W}_y (\Delta t) ; \\
S_z &= \Delta \tilde{X}_3 = F_z \Delta t + G_{zz} \tilde{W}_z (\Delta t) .
\end{align*}
\]

(6-12)

Discussion

(1) As was pointed out in the preceding section, the advection-diffusion equation cannot be applied to describe diffusion in turbulent flows unless the size of the solute cloud is large with respect to the Lagrangian integral length-scale of turbulence. As was shown by Taylor (1921) in his classical analysis of turbulent diffusion of particles, the variance of the displacements of the particles is related to the autocorrelation function of the particle velocities. For particles released at a fixed point, two stages of diffusion can be roughly distinguished after the release. For a time small compared to the Lagrangian integral time-scale of turbulence, the particle motions are fully correlated so that the variance increases as a quadratic function of time. After this stage the particle motions become more and more uncorrelated so that eventually the variance increases linearly with time. The advection-diffusion equation can then be applied to describe the probability density function of the displacements of the particles, with a constant eddy diffusivity (see equations (6-6), (6-7) and
(6-10)). This stage of diffusion is also characterized by the size of the solute cloud which is much larger than the Lagrangian integral length-scale of turbulence. In shallow water flows this length-scale is of the order of the water depth. Since the horizontal dimensions of a solute cloud are usually much larger than the water depth, the advection-diffusion is then applicable. There are however exceptions to this general rule, see section 6.1.

(2) It is shown above that the theory of the random walk method is based on the analogy between the Fokker-Planck equation and the advection-diffusion equation. Correspondences between the parameters of the random walk and the flow quantities are derived through the analogy. Therefore, as far as the theoretical formulations are concerned, the so-called Lagrangian approach based on the particle tracking method is an analogous solution of dispersion described by the advection-diffusion equation.

(3) The term $\partial D_i/\partial X_j$ in equations (6-10) comes quite unexpectedly, as intuitively one would expect $F_i$ to correspond to the advective fluid velocity only. This term is in fact a correction for the nonhomogeneity of turbulence that otherwise would cause accumulation of particles at locations in the flow field with lower diffusivity (Uffink, 1990; see also section 6.3.2). The Itô convention is used in the present model. Similar correction terms are obtained when the Stratonovich convention is used.

(4) When implementing the particle method on digital computers, a limited number of particles can be used to represent the solute mass, which are traced in the fluid flow by using equations (6-1) and (6-10). The concentration is then approximated by the distribution of these particles. The random walks are realized with a finite time-step $\Delta t$. Inaccuracies are thereby introduced in the obtained concentration distribution. On the other hand, a large number of particles and a small time-step are attended with a large computational effort. One needs to find for each specific problem an appropriate time-step and number of particles to achieve the desired accuracy of the concentration distribution, at acceptable computational cost.
Boundary conditions

Two types of boundaries are considered in the present particle model: a discharge point and an impermeable surface. A discharge point is modelled by releasing at that point finite numbers of particles successively at each time-step according to the prescribed time-history of the discharge rate. At an impermeable boundary surface the transport rate of matter in the direction perpendicular to that surface is equal to zero. This is realized in the particle model by using complete reflection of particles at the surface. Examples of such boundaries are the bottom, the free surface and solid walls.

6.2.2. General structure of the model

The numerical model of dispersion of solutes in turbulent shear flow developed in the present study is based on the theory presented in the preceding section. The model uses flow data computed by the Q3D flow model DUTRID (see figure 6.1).

The computation proceeds as follows. The initial concentration distribution is first transformed into discrete particle masses in each grid-cell. The coordinates of all particles are stored in a large array. Then for each step of tracking, each particle is displaced following pure advection (see section 6.4 for details), followed by a random displacement that simulates turbulent diffusion (see section 6.3). During these steps of tracking, new particles are introduced into the fluid flow at discharging points of solute (if there are any). At the same time, particles exiting the flow together with water outflows are removed from the coordinate array and thereby excluded from subsequent steps of particle tracking. The discretized concentration distribution at any instant is computed as the total particle mass inside each grid-cell, divided by the volume of that grid-cell.

The time-step of particle tracking is to be prescribed by the user. Combined with the diffusivity, it determines the lengths of maximum random displacements of the particles. An upper limit must be imposed to these lengths in order that the dispersion because of velocity shear in vertical planes is well simulated. An obvious choice for this limit is a typical vertical grid distance. The resulting time-step is usually quite small for typical turbulence found in
shallow water flow. As a rule, it is much smaller than the time-step of computation used in the flow model (see section 6.5.3).

The general structure of the particle model is illustrated in figure 6.1.

6.3. Modelling turbulent diffusion

Except during the initial stage of mixing after a solute discharge, the dispersion of the solute in shallow waters is essentially an equilibrated process of longitudinal stretching of clouds of solute caused by vertical velocity shear, and transverse diffusion (see section 6.5). It is therefore essential to accurately predict the turbulent diffusion process in homogeneous as well as nonhomogeneous turbulence. Uniform flows are assumed in this section in order to concentrate on the modelling of diffusion. Particle tracking in nonuniform flow is described in section 6.4.

In the particle tracking model turbulent diffusion is simulated by prescribing random
displacements of the particles. The magnitudes of these displacements are related to the turbulent diffusivity or eddy viscosity of the flow (equations (6-11)). The way the random walks are realized on a computer is described below. The effects of nonhomogeneity of the turbulence upon computed concentration distributions are examined in section 6.3.2 using a computational example.

6.3.1. Homogenous turbulence

Diffusion of a solute in uniform flow with homogenous turbulence is obviously similar to the classical case of molecular (or Fickian) diffusion in a quiescent medium. The well-known solution of Fickian diffusion is then applicable for the case under consideration when the molecular diffusivity is replaced by the eddy diffusivity. Only one-dimensional diffusion is considered here for simplicity. The analysis can easily be extended to two- and three-dimensional cases.

We first introduce a series of independent real random variables \( \mathcal{S}(i) \) \( (i = 1, 2, \ldots) \). Each of these variables has a uniform distribution over the interval \([S_m - S_v, S_m + S_v]\), where \( S_m \) and \( S_v \) are two real numbers to be defined later. This distribution can be obtained by using a random floating-point number generator available on most computers. One can easily derive the mean \( (E_s) \) and standard deviation \( (\sigma_s) \) of \( \mathcal{S} \), the results being as follows:

\[
E_s = \langle \mathcal{S} \rangle = S_m ;
\]

\[
\sigma_s^2 = \langle \mathcal{S}^2 \rangle - E_s^2 = \frac{S_v^2}{3} . \tag{6-13}
\]

Suppose a particle is at \( x = 0 \) at time \( t_o = 0 \). It undergoes for time \( t = t_o, t_1, \ldots, t_n \) successive independent random displacements with step \( \mathcal{S}(i) \) \( (i = 1, 2, \ldots, n) \). According to the central-limit theorem, the probability density distribution \( P(x, t_n) \) of the particle location at time \( t_n \) approaches a Gaussian distribution as \( n \) increases:

\[
P(x, t_n) \approx \frac{1}{\sqrt{2\pi n \sigma_s^2}} \exp \left(-\frac{(x - nE_s)^2}{2n\sigma_s^2}\right) . \tag{6-14}
\]

The concentration distribution \( c(x, t) \) of a solute of unit mass released instantaneously at \( t = 0, x = 0 \) in a uniform flow with a velocity \( u \) and diffusivity \( D_m \) is well known:
\begin{equation}
\frac{c(x,t_n)}{l_n} = \frac{1}{\sqrt{4\pi D_n t_n}} \exp \left( -\frac{(x - u t_n)^2}{4 D_n t_n} \right). \tag{6-15}
\end{equation}

The analogy between equations (6-14) and (6-15) is obvious. Since \( t_n = n \Delta t \), equating corresponding terms in both equations yields \( E_x = u \Delta t \); \( \sigma_x^2 = 2D_m \Delta t \), or, using equations (6-13):

\begin{align*}
S_m &= u \Delta t; \\
S_v &= \sqrt{6D_m \Delta t}. \tag{6-16}
\end{align*}

Note that equations (6-16) can also be directly obtained by using the general relations derived in the preceding section. Substitution of (6-11) into (6-4) yields:

\( <S(i)> = u \Delta t; \quad <S(i) S(i)> - (u \Delta t)^2 = 2D_m \Delta t. \)

Substituting these relations into equations (6-13), one obtains equations (6-16).

Using the particle model developed in the present study, a numerical experiment was made simulating one-dimensional turbulent diffusion in still water between two vertical walls.
A unit mass of solute was represented by a total number of 10,000 particles, discharged instantaneously at \( x = 5 \) m at time \( t = 0 \). Particles reaching the vertical walls at \( x = X_i \) and \( x = -X_i \) are perfectly reflected. The following quantities were assumed in the computations:

\[
D_m = 0.01 \text{ m}^2/\text{s}; \quad \Delta t = 10 \text{ s}; \quad X_i = 500 \text{ m}.
\]

The computed concentrations are shown in figure 6.2 for \( t = 100 \) s, 200 s and 500 s after discharge of the solute. No particle reached a vertical wall in the simulations presented here. The numerical results of concentrations are obtained by calculating the total mass of particles found within each interval of 0.5 m. The Gaussian distributions given by (6-15) are also plotted in this figure. The agreement between the numerical results and the analytical solutions is quite good. Test computations using a smaller time-step (equal to 1 s) showed little difference in the results.

6.3.2. Nonhomogenous turbulence

In the computational example presented in the preceding section a constant diffusivity was assumed. In practical situations, however, the turbulent diffusivity or eddy viscosity nearly always varies in space. For example, the vertical turbulent diffusivity in shallow flows shows a more or less parabolic distribution in the vertical direction (see chapter 4). When diffusion in nonhomogeneous turbulence is considered, equation (6-16) that prescribes the random walk for homogenous turbulence, would yield unrealistic accumulation of particles at locations with smaller diffusivity. Therefore, advective corrections (Uffink, 1990; also see discussion (1) in section 6.2.1) must be made to overcome this difficulty, as will be demonstrated by the computational example below. Uffink examined the importance of this correction term for diffusion under similar conditions.

Using the particle tracking model, the one-dimensional horizontal diffusion in a quiescent medium between two solid walls was simulated. The walls are located at \( x = 0 \) m and \( x = 10 \) m, respectively. At \( t = 0 \) the solute is uniformly distributed. The diffusivity is assumed to be a parabolic function of \( x \). Its maximum value is 0.01 m\(^2\)/s at \( x = 5 \) m and approaches zero at the solid walls. 5000 particles were used in the simulations, representing one kilogram of solute mass.
The following relations should be used for the random walk (c.f. equations (6-11)):

\[
S_m = \left( u + \frac{\partial D_m}{\partial x} \right) \Delta t ;
\]

\[
S_v = \sqrt{6D_m \Delta t}.
\]  

(6-17)

Note the correction term \( \frac{\partial D_m}{\partial x} \) in the expression of \( S_m \) (c.f. equations (6-16)). The computed concentrations 200 s after the solute discharge are shown in figure 6.3 with the correction term either being included or neglected. A uniform particle distribution along \( x \) is obviously expected as the correct solution. Unrealistic accumulation of particles at the solid walls, where the diffusivity is minimum, results if no correction is made in \( S_m \).

The accumulation of particles at the solid walls arises as follows. Following the expression of \( S_v \) in equations (6-16) or (6-17), particles located at points with larger diffusivity (in the central part of the axle) make larger random jumps at each step of tracking than those located at points with low diffusivity (near the walls). An artificial net flux of particles from the centre towards the walls is thereby created. It is this flux, if not balanced

![Figure 6.3. Diffusion in nonhomogeneous turbulence.](image)

(a) correction term excluded; (b) correction term included.
by the correction term, that causes particles to accumulate near the walls.

6.4. Modelling advection

Solute particles are displaced in a fluid flow by the time-mean current. This is known as advection, and is the most important part of the dispersion process. In a particle tracking model, advection is modelled through the deterministic displacements of the particles (see equation (6-1)). These displacements also include the corrections for the nonuniformity of the eddy diffusivity (see equations (6-10)).

In this section the cell-analytical method used for modelling advection in the particle model, is presented. By using this method the computed particle trajectories in any two-dimensional steady-state flow in a closed basin remain closed. The model is verified for two-dimensional advection in a rigidly rotating flow field (the so-called Molenkamp test). The used modelling method is compared with a conventional numerical integration method for this test case.

6.4.1. Cell-analytical solution method

The cell-analytical solution method adopted in the particle model is presented for advection in two dimensions. Extension to three dimensions is straightforward.

Assuming a two-dimensional flow with zero diffusivity, the movement of a particle in that flow satisfies the following equations:

$$\frac{dx}{dt} = u(x,y,t) ;$$

$$\frac{dy}{dt} = v(x,y,t) .$$ (6-18)

This set of ordinary differential equations is to be solved for a given velocity distribution $u, v(x,y,t)$. The conventional method of solving equations (6-18) is by using numerical integration. This method is easy to implement in a numerical model. However, the truncation errors, which are unavoidable with this method even when higher-order schemes
are used, result in artificial net drift of particles or a solute cloud. This drift is particularly undesirable because it accumulates with time (see example in the next section). One can use a smaller time-step to achieve higher accuracy, but the numerical efficiency of the particle model is then affected. Practical applications of a dispersion model usually involve long periods of time. A dispersion model is hardly useful for such applications unless it is numerically efficient.

![Figure 6.4. Configuration of a grid cell used for particle tracking.](image)

In order to obtain accurate predictions of the advection of particles, an alternative to the conventional approach was devised (Stelling, 1991, personal communication). Using this method equations (6-18) are solved analytically per grid-cell. Since the solution of the trajectories of particles is analytical, drift of particles typical of the conventional method does not occur, as will be illustrated later in this section.

Since the particle model was designed to run in conjunction with the flow model DUTFID, particle tracking is made with reference to the same finite-difference grid as used in DUTFID (see figures 3.1 and 3.2). The tracking cells are centred around the H-points in the staggered grid, see figure 6.4.

Let $u_r$, $u_1$, $v_1$ and $v_r$ represent the velocity components at the centres of the four sides of the tracking cell, see figure 6.4. Using a local coordinate system, the velocity distribution in the cell can be approximated using linear interpolations:
\[
\frac{dx}{dt} = u(x, y, t) = u_i + \frac{x}{L}(u_r - u_i), \\
\frac{dy}{dt} = v(x, y, t) = v_i + \frac{y}{B}(v_r - v_i);
\]

where \(L\) and \(B\) are the lengths of the cell in \(x\)- and \(y\)-directions, respectively. Assuming an initial condition \(x = x_o, y = y_o\) at \(t = 0\), the solution for (6-19) can easily be found. In case \(u_i \neq u_r\) and \(v_i \neq v_r\), we have

\[
x = (x_o + \xi) \exp \left( \frac{(u_r - u_i) t}{L} \right) - \xi,
\]

(6-20)

\[
y = (y_o + \eta) \exp \left( \frac{(v_r - v_i) t}{B} \right) - \eta;
\]

where

\[
\xi = \frac{u_i L}{u_r - u_i}; \quad \eta = \frac{v_i B}{v_r - v_i}.
\]

(6-21)

If \(u_i = u_r\) the solution for \(x\) is \(x = x_o + u_i t\). If \(v_i = v_r\) the solution for \(y\) is \(y = y_o + v_i t\).

One can verify that the assumed velocity distribution in the grid-cell satisfies the equation of continuity. Using equations (6-19), the equation of continuity becomes

\[
\frac{u_r - u_i}{L} + \frac{v_r - v_i}{B} = 0.
\]

(6-22)

This equation is also the central-difference approximation of the continuity equation at the H-point in the staggered grid, used in the flow model. The divergence of velocity is therefore equal to zero for the assumed velocity distribution in all grid-cells, and a divergence-free velocity distribution is then obtained over the whole flow domain.

The theory of the cell-analytical method presented above is rather simple. However, its operation with a numerical model seems to require no less effort than the conventional method. Several steps are followed for one time-step of advection of one particle. First the current position of the particle is transformed into grid numbers and the local coordinates in the corresponding grid-cell. Then velocity components at the four sides of the cell \(u_i, u_r, v_i,\)
and \( \nu \), are interpolated by using the velocities at the H-points in the staggered grid, computed in DUTRID (see figure 6.4). Subsequently, by computing \( \xi \) and \( \eta \) (equation (6-21)), it is judged towards which two sides of the cell the particle is moving. The times for the particle to reach those sides are computed by using equations (6-20). The smaller of the two computed times is the time the particle needs to move across the cell, with which the end-position of the particle on the cell sides is computed. In case this time is smaller than the time-step used for tracking, tracking continues with the grid-cell into which the particle moves.

Checking of the position of each particle must be made at each time-step during the entire tracking procedure. A particle may be located exactly at the interface of two grid-cells or even at the point of intersection of two such interfaces. It is then necessary to check which cell the particle is moving into. The local velocity needs to be interpolated and used for that purpose. The position of each particle also needs to be checked against boundaries of the flow field.

The more conventional method of solving equations (6-18) by using numerical integration is not implemented in the particle tracking model. It was nevertheless used to simulate simple advection in a two-dimensional rotating flow, and compared to the cell-analytical method (see next section).

6.4.2. The Molenkamp test

In the Molenkamp test a solute cloud is advected in a rigidly rotating flow with its centre within the flow domain. After each period of rotation the solute cloud should obviously return to its original position without change of concentration distribution. The test is often used for testing numerical schemes used to solve the advection equation written as follows (Molenkamp, 1967; Van Stijn, 1988)

\[
\frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} + v \frac{\partial c}{\partial y} = 0 .
\]

The Molenkamp test can also be used to examine the accuracy of the method of modelling advective transport using a particle model. Since numerical diffusion is absent in
Figure 6.5. Molenkamp test, trajectory of a particle.

the model, sharp gradients of concentration are preserved. It suffices to use one particle for
the test, and to examine the final position of the particle after several rounds of rotation, with
respect to its original position. The results of such a numerical simulation are presented
below.

The flow domain used for the simulation is represented by $11 \times 11$ grid points (see
figure 6.5). The origin of coordinates is chosen at the centre. For a rigidly rotating flow
around the centre the velocity at each grid point is then equal to $(-\omega y(i,j), \omega x(i,j))$ where
$\omega$ is the angular speed of rotation and $x(i,j), y(i,j)$ are the coordinates of grid point $(i,j)$. At
$t = 0$ the particle is located on the $y$-axis at some distance from the centre of rotation.
Particle tracking commences at $t = 0$ and continues for 200 periods of rotation. The
computed trajectory of the particle is plotted in figure 6.5. It is seen to be closed.

For comparison with the cell-analytical solution method, a conventional numerical
integration method was used for the same simulation. The commonly used fourth-order
Runge-Kutta algorithm was used to solve the trajectory of the particle.

The computed results depend markedly upon the time-step used in the numerical
integration. The trajectory of the particle shown in figure 6.6 was obtained using a time-step approximately equal to $L/U$, where $U$ is the orbital speed of the particle; $L$ is the grid-
spacing. Particle tracking continued for 200 periods of rotation. A net drift towards the rotation centre can be observed.

The results presented above show that with the cell-analytical approach the net drift of particles, which arises when the numerical integration is used, can be reduced. Nevertheless, errors caused by grid discretization of the flow domain as well as the linear interpolation of velocity distributions in grid-cells, cannot be avoided. These errors seem to result in an inaccuracy in the predicted period of rotation of a particle. The trajectories of several particles for precisely one period of rotation are plotted in figure 6.7. It is seen that all particles have not returned to their original positions, although the particles do reach their original positions when tracking continues.

6.4.3. Advection in vertical planes

In DUTRID an equal number of levels is used in each vertical to solve the vertical distribution of velocities, see figures 3.2 and 3.3. Similarly, an equal number of cells in all verticals is used for particle tracking. The bottom and the top surfaces are assumed horizontal within each cell, see figure 6.8. The velocities at the centres of cell walls ($u_l$, $u_r$, ... in figure 6.8) are obtained using linear interpolation. This simple configuration was chosen for two reasons. Firstly, it is consistent with the configuration in the horizontal plane (c.f. figure

![Figure 6.8. Grid cell configuration in the vertical plane.](image-url)
6.4) so that the same computation procedure can be followed for particle tracking in all three dimensions. Secondly, considering the efficiency of the particle tracking procedure, it is important that the time for a particle to reach a cell boundary can be computed explicitly. Other configurations of the grid-cell, which may fit the cell boundaries better, lead to more complicated solutions of the particle trajectory, and iteration would be needed to calculate the time of arrival of a particle at a cell boundary.

The velocities perpendicular to the bottom and the free surfaces are assumed equal to zero so that particles can never cross these surfaces. Since the vertical grid-spacing can be different for each vertical because of the varying water-depth, the vertical coordinates of particles are used for tracking rather than the cell numbers used for horizontal displacements.

6.5. Two-dimensional shear dispersion

In practical situations dispersion always results from the combined actions of advective current and turbulent diffusion. In this section we examine shear dispersion in a two-dimensional flow, see figure 6.9. The velocity distribution is assumed uniform in $x$-direction and arbitrary in the vertical direction. Solute mass is injected uniformly across the water depth at $x = 0$ at time $t = 0$. A coordinate system is chosen such that $H_s = 0$.

After injection the solute cloud is quickly stretched in $x$-direction because of velocity shear, while turbulent diffusion in the vertical direction gradually spreads the solute cloud over the entire water depth. At a later stage of mixing, the cloud of solute is stretched over a long distance in the longitudinal direction, and the concentration distribution becomes nearly uniform in the transverse direction. The longitudinal distribution of the depth averaged concentration approaches a Gaussian distribution, as was proved by both theoretical analysis and experimental evidence (Fischer et al, 1979).

The theory of turbulent shear dispersion presented in the classical work of Taylor (1954) is briefly outlined in the next section. After that, results of numerical simulations using the particle model are presented (section 6.5.2). The specific case of modelling dispersion in shallow waters is discussed in section 6.5.3.
6.5.1. Theory

The basic theory of dispersion in a steady uniform turbulent shear flow was developed by Taylor (1954). It has been applied for studying dispersion in a wide variety of flows, such as in rivers and estuaries (Fischer et al, 1979).

Several quantities need to be defined for the following analysis, see figure 6.9. The depth averaged flow velocity is:

$$\bar{u}(x,t) = \frac{1}{D} \int_0^D u(x,z,t) \, dz.$$  

The deviation of the velocity from this average is $u''(x,z,t) = u(x,z,t) - \bar{u}(x,t)$.

Similarly, the depth averaged concentration is:

$$\bar{c}(x,t) = \frac{1}{D} \int_0^D c(x,z,t) \, dz.$$  

The deviation of the concentration from this average is $c''(x,z,t) = c(x,z,t) - \bar{c}(x,t)$. Using these definitions, and neglecting horizontal diffusion, the depth-averaged advection-diffusion equation becomes:

$$\frac{\partial \bar{c}}{\partial t} + \bar{u} \frac{\partial \bar{c}}{\partial x} + \frac{1}{D} \frac{\partial}{\partial x} \left[ \int_0^D u'' \, c'' \, dz \right] = 0 .$$  

(6-23)
The integral in the above equation represents the depth-integrated transport rate ($\Omega$) caused by velocity shear. According to Taylor (1954):

$$\Omega = \int_0^D u'' c'' \, dz = \frac{\partial c}{\partial x} \int_0^D u'' \, dz \int_0^z \frac{1}{D_m} \, dz' \int_0^{z'} u'' \, dz'' .$$  \hspace{1cm} (6-24)

This equation shows that the transport rate $\Omega$ is proportional to the gradient of the mean concentration in the longitudinal direction. This is similar to Fickian diffusion where the proportionality constant is defined as the molecular diffusivity. A *longitudinal dispersion coefficient* ($K$) can then be defined as follows:

$$\Omega = -D \, K \frac{\partial c}{\partial x} ,$$  \hspace{1cm} (6-25)

in which

$$K = -\frac{1}{D} \int_0^D u'' \, dz \int_0^z \frac{1}{D_m} \, dz' \int_0^{z'} u'' \, dz'' .$$  \hspace{1cm} (6-26)

The depth-averaged advection-diffusion equation then becomes:

$$\frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} = K \frac{\partial^2 c}{\partial x^2} .$$

The solution of this equation in an unbounded domain for an initial unit pulse at $x = 0$ is a Gaussian distribution with a variance ($\sigma^2$) growing at a constant rate (c.f. equation (6-15)):

$$\frac{d\sigma^2}{dt} = 2K$$  \hspace{1cm} (6-27)

For uniform free-surface flow over a bed with a constant slope, logarithmic and parabolic distributions over the depth can be assumed for the velocity difference and the eddy diffusivity, respectively (see section 5.2):

$$u'' = \frac{u^*}{\kappa} \left[ 1 + \ln \frac{z}{D} \right] ;$$  \hspace{1cm} (6-28)

$$D_m = \kappa u^* z \left[ 1 - \frac{z}{D} \right] .$$

Substituting of (6-28) into (6-26) yields the well-known result (Elder, 1959):

$$K = 5.93 \, D \, u^*. $$  \hspace{1cm} (6-29)
Note that the various assumptions made in the above analysis are valid only at a time
$t_i$ sufficiently long after the solute discharge. $t_i$ is expressed as fraction ($\alpha$) of the time-scale
($t_L$) for the cross-sectional mixing:

$$t_i = \alpha t_L$$    \hspace{1cm} (6-30)

where, for a constant diffusivity $D_m$, $t_L$ is defined as below:

$$t_L = \frac{D^2}{D_m}.$$    \hspace{1cm} (6-31)

For a parabolic distribution of $D_m$ (equations (6-28)), one may use the depth-averaged
value in (6-31). Chatwin (1970) showed that the longitudinal distribution of the average
concentration $\bar{c}$ approaches a Gaussian distribution for $\alpha \geq 0.4$. A number of numerical
studies have shown that the variance of the dispersing cloud grows at a constant rate for $\alpha$
$\geq 0.2$ (Bugliarello and Jackson, 1964; Fischer, 1968).

6.5.2. A numerical example

An analytical solution is available for two-dimensional turbulent shear dispersion, as
was presented in the preceding section. This interesting case of dispersion can be used to
verify numerical dispersion models. The particle model was used in this section to simulate
one case of two-dimensional shear dispersion. The results are compared with the analytical
solution.

The following quantities were assumed in the numerical simulations:

$$D = 10 \text{ m}; \quad u^* = 0.05 \text{ m/s}; \quad z_0 = 0.01 \text{ m}; \quad N = 4000;$$

where $N$ is the number of particles used. The velocity profile and the eddy diffusivity were
computed according to equations (6-28).

The variance of the depth-averaged concentration (or particle distribution) in the flow
direction was computed from

$$\sigma^2 = \frac{1}{N} \sum_{i=1}^{N} (x(i) - \bar{x})^2, \quad \text{where} \quad \bar{x} = \frac{1}{N} \sum_{i=1}^{N} x(i).$$

Computations were made with 5, 10, 20 and 40 levels in the vertical. Velocities at the
grid points were computed according to the assumed velocity distribution. The time-step used
for particle tracking was 2 s. Initially, the particles were uniformly distributed over the water depth. Particle tracking began at $t = 0$ and continued until $t = 400$ s. The numerical results of the variance $\sigma^2$ are shown in figure 6.10. It is seen that the computed rates of increase of $\sigma^2$ approach the analytical solution (equation (6-27)) when more grid levels are used in the simulations. Note that the theory only predicts the rate of change of $\sigma^2$, not the value itself.

The initial stage of dispersion after solute discharge can be observed in the numerical results. At this stage the variance increases at a smaller rate. According to the result obtained using 40 levels in the vertical, the linear increase of variance with time begins approximately 400 s after solute discharge. The time-scale $t_e$ for the cross-sectional mixing is equal to 2000 s for the flow considered, which corresponds to $\alpha = 0.2$ (see equation (6-30)). This is in good agreement with Bugliarello and Jackson (1964), and Fischer (1968).
6.5.3. Shear dispersion in shallow water flow

When modelling dispersion of a solute in flows, two different physical processes must be considered, namely advective transport and turbulent diffusion. The accuracy of prediction of a dispersion model depends on those achieved when modelling these two processes. As far as the particle model is concerned, the time-step used for particle tracking is an important factor (see section 6.2.1, discussion (3)).

Turbulent diffusion is simulated in the particle model by using random displacement of particles. The maximum length of the random jump is proportional to the square-root of the time-step (see equations (6-16)). When a large time-step is used, particles may go across several grid-cells with one single random jump. In case the vertical gradients of flow velocity and eddy viscosity are large, as is the case with shallow water flows, this would result in large inaccuracy in the predicted displacement of the particles. As a rule, one should choose a tracking time-step such that the random jump of a particle at most covers a length equal to the cell size. Therefore, the time-step is restricted by the vertical grid-spacing used.

The advective transport is resolved in the particle model by using an analytical approximation within each grid-cell (see section 6.4). Obviously, the maximum allowable time-step for modelling advection is set by the time a particle needs to cover a length equal to the cell size with the advective current.

Particle tracking in shallow water is characterized by a large length (of the order of one kilometre) to height (of the order of one meter) ratio of the tracking cell. Therefore, as far as turbulent diffusion is concerned, a much larger time-step can be used in the horizontal plane than in the vertical direction. The advective transport, on the other hand, takes place predominantly in the horizontal directions. The vertical advective velocity is usually quite small. Due to the large horizontal grid-spacing, a relatively large time-step can be used when modelling advection. As a result, as far as the maximum allowable time-step used in the particle model is concerned, modelling turbulent diffusion in the vertical direction is the most restrictive part.

One could therefore choose a time-step according to the condition that the maximum length of random jump is less than the vertical grid-spacing. For practical applications,
however, this condition appears to be too restrictive, as the obtained tracking time-step can be very small. The computational efficiency of the particle model is thereby affected, especially because a large number of particles is usually required. This becomes more evident when making a rough estimation of the maximum allowable time-steps for modelling advection and turbulent diffusion. In lakes and reservoirs the water velocity is of the order of 10 cm/s. For a horizontal grid-spacing of one kilometre, it takes 10,000 s for a particle to travel horizontally across one grid-cell. A typical value of 0.01 m/s can be assumed for the velocity scale \( u^* \) used for estimating the vertical eddy diffusivity. For a water depth of 10 m and assuming a parabolic distribution of the vertical eddy diffusivity, the average diffusivity is then equal to \( \kappa u^* D/6 \) or 0.00667 m\(^2\)/s. The vertical grid-spacing is equal to 1 m when 10 levels are used. Using equations (6-16), the time-step corresponding to a maximum length of the random jump equal to 1 m, is approximately 25 s.

Because of the large difference between the time-scales involved in modelling diffusive and advective transports, it is useful to examine how the accuracy of the predicted concentration distribution actually depends upon the time-step used. In case of a weak dependence one could reduce the CPU time by using a larger time-step. This is particularly helpful from the point of view of applications.

The possibility of using a larger time-step when simulating dispersion in shallow water flows was investigated by using the computational example presented in the preceding section. Using 10 levels in the vertical, computations were made using different time-steps for particle tracking. The lower and upper limits of the time-step \( t_1 \) and \( t_2 \) were chosen such that the maximum length of a random jump was equal to the vertical grid spacing \( \Delta D \) and to the water depth, respectively. Using the maximum value of the vertical eddy diffusivity \( D_m \) in equations (6-16), we have:

\[
\begin{align*}
    t_1 &= \frac{\Delta D^2}{6 D_{\text{max}}} , \\
    t_2 &= \frac{D^2}{6 D_{\text{max}}} ;
\end{align*}
\]

and the various time-steps are determined according to

\[
\Delta t = t_1 + \alpha (t_2 - t_1) ; \quad \alpha = 0 \sim 1
\]

Results for the variance of particle distributions in the flow direction are shown in
Figure 6.11. Two-dimensional shear dispersion in shallow water flows. (a) through (f): computations for $\alpha = 1.0, 0.5, 0.3, 0.2, 0.1$ and 0.05, respectively.

Figure 6.11 for 6 different values of $\alpha$. The results show little difference for $\alpha < 0.3$, which indicates a weak dependence of computed particle distribution upon the time-step used for particle tracking. The difference between the numerical result corresponding to $\alpha < 0.3$ and the analytical solution can be attributed to the small number (10) of vertical levels used in the simulations (c.f. figure 6.10).

The ratio of upper to lower limit of the time-step $t_u/t_l$ is equal to 100 in this example. In other words, the total number of steps of computations is approximately 100 times less when $\alpha = 1$ than when $\alpha = 0$. The relative error of computed variances at time $4 \times 10^4$ s is for $\alpha = 1$ approximately 20% with respect to the solution for $\alpha = 0$. In one-dimensional dispersion the concentration is approximately inversely proportional to the length over which the particles are distributed. This length is of the order of the square root of the variance. The corresponding relative error of computed concentrations is therefore about 10%, which could be considered as within the acceptable limits for applications.
6.6 Summary and conclusions

Based on the random walk theory, a particle tracking model of dispersion of passive dissolved matter in turbulent shear flows has been developed. The particle model runs in conjunction with the numerical flow model DUTRID. The particle method is based on the analogy between the Fokker-Planck equation of random walk and the advection-diffusion equation.

In the particle model, advection is simulated by using a cell-analytical approach. Net drift of particles which is typical of conventional numerical integration methods, is thereby reduced. This was illustrated for pure advection of a local concentration in a rigidly rotating flow (the Molenkamp test).

Turbulent diffusion is simulated by adding a random component to the advective displacement of particles. A random number generator available on most computers is used. The magnitude of the random component is related to the eddy diffusivity. Examples simulating one-dimensional diffusion in homogeneous and non-homogeneous turbulence were given.

Two-dimensional shear dispersion, which to a large extent approximates dispersion under practical circumstances, was simulated using the particle model. Computations were made with various numbers of vertical grid points. The variance of the particle distribution in the flow direction was computed as a function of time and compared with the analytical solutions given by Taylor (1954) and Elder (1959). Good agreement was obtained for the rate of increase of the variance.

The accuracy of a particle model depends on the number of particles and the time-step used for particle tracking. The time-step is limited by the maximum allowed length of random jump of particles. Particle tracking in shallow water flows is characterized by a large length to height ratio of the tracking cell. Accurate simulation of turbulent diffusion in the vertical direction usually requires a very small time-step. The influence of the time-step upon
the concentration distribution was investigated using a numerical example. The results showed that the maximum allowed length of random jump can be as large as the water depth. The resulted error in predicted concentrations was within acceptable limits for engineering applications.
CHAPTER 7

MODELLING DISPERSION OF A SOLUTE IN LAKE YSSEL

7.1 Introduction

Lake Yssel is an artificial lake in the north of the Netherlands created by damming an inland tidal sea in the historical process of land reclamation (see figure 7.1). It has a surface area of approximately 1120 km², with an average water depth of about 5 m. Being the largest inland water in the Netherlands, it has significant ecological and economical interests.

Lake Yssel provides an important supply for drinking-water and water for agricultural purposes in the lake region. In order to assure safe and reliable water supply under quality controlled conditions, various studies on the hydrodynamics and the dispersive characteristics of the lake have been carried out at the initiative of the Dutch government and some private water companies (RIWA, 1990). Both computer models and field measurements are used in these studies. Topographical and hydrological conditions have been surveyed and used in the computer models.

At the initiative of the N.V. Watertransportmaatschappij Rijn-Kennemerland, a waterworks company, and Rijkswaterstaat-RIZA, Dutch Ministry of Transport and Public Works, a campaign of field measurements of concentration distributions of a solute in Lake Yssel was carried out in 1990. The solute (Rhodamine) came from the River Yssel which discharges into the lake, and gradually dispersed over the whole lake area. Measurements of concentrations continued for a period of nearly two months, starting at the end of September. They were made on board a measuring vessel regularly traversing the solute cloud area.

Lake Yssel is a typical shallow lake to which the flow and dispersion models developed
in the present study can be applied. These models were used to simulate the above-mentioned field experiment with the aim to test these models for practical applications. Values of model parameters and input variables were taken from the measurements when possible. Other ones were determined using empirical relations. A sensitivity study was made regarding the dependence of the computed concentrations upon these parameters and variables.

Results of these numerical simulations are presented in this chapter. In section 7.2 details of the field measurements are presented. The hydrodynamic conditions of Lake Yssel such as the horizontal geometry, bottom topography, inflows and outflows, and wind conditions are also described. A study of the hydrodynamics of the lake is presented in section 7.3 for both steady wind conditions and the variable wind recorded in the experiment. Simulations of the dispersion of solute are described in section 7.4. Computational results of solute concentration distributions are shown and compared with the measurements. Some
conclusions and a discussion are presented in section 7.5.

7.2 Field experiment in Lake Yssel

7.2.1 General description of the experiment

Under the assignment of the Dutch Ministry of Transport and Public Works, an experimental study on the dispersion of matter in the River Rhine was made in September, 1990. Part of the solute (Rhodamine) released in this experiment reached the River Yssel and was discharged into Lake Yssel.

The solute first entered the Ketelmeer, a wide channel connecting the river to the main lake (see figure 7.1). Measurement of solute concentrations began on 1 October when the solute cloud first appeared in the main lake. Measurements were made on board a measuring vessel traversing the lake. While samples of lake water were taken, the instantaneous positions of the vessel were monitored by means of a satellite navigation system. The accuracy of this position-finding system is 30 to 100 m. The concentrations of Rhodamine in the samples were measured in a laboratory. The surveys took place at intervals of two or three days in the beginning, then at intervals of about one week at a later stage. Observations on each day began around 9 a.m. and continued until about 3 or 4 p.m.

Rhodamine is a reddish passive chemical, soluble in water. The concentration of Rhodamine in water can be measured by using, among other methods, High Pressure Light Chromatography (the HPLC method). This method is suitable for measuring relatively low concentrations, though the accuracy is then low. It was used in this experiment because of the low concentrations encountered (of the order of 1 μg/m³).

Discharge of solute

During the experiment the concentrations of Rhodamine were continually observed at Kampen, a small town near the mouth of the River Yssel, and at Ketelbrug where the
*Ketelmeer* joins the main lake. Observations are shown in figure 7.2.

It is seen that concentration peaks arrived soon after the first concentrations were observed. Afterwards, the concentration first decreased rapidly, then much more gradually, showing a long tail which was observed during about one week. Little differences were found between observations made at three different locations across the river, which indicates that the solute was well mixed across the river section. The width of the river at *Kampen* is about 100 m.

On the other hand, marked variations of the concentration were found in the cross-

Figure 7.2. Measurements of solute concentrations,  
(a) at Kampen; (b) at Ketelbrug (Note difference in scales).
section at Ketelbrug, which suggests that the solute was not yet fully mixed in the transverse direction of the channel when reaching the main lake. The maximum width of the Ketelmeer is about 4 km; its smallest width is found at Ketelbrug (approximately 1 km).

7.2.2 Hydrodynamic conditions during the experiment

General characteristics of Lake Yssel, such as the horizontal geometry, bottom topography, inflows and outflows, and wind conditions, are described in this section. They were made available by Rijkswaterstaat-RIZA where numerical simulations of the field experiment were made using depth-averaged flow and dispersion models. These data are used in the next section where the simulation of the flow in the lake is presented.

*Horizontal geometry, bottom topography*

Partly surrounded by reclaimed land and straight dams, Lake Yssel has mostly straight lateral boundaries (see figure 7.1). A dam approximately 25 km long, situated at the north end of the lake, separates the lake from the shallow tidal waters behind the Frisian islands (Wadden Sea). At each end the dam contains discharge and navigation sluices. A second dam, at the southern half of the lake and running in the longitudinal direction, divides the lake into two completely isolated parts. The northern part, which is considered in this study, is approximately 50 km long and 25 km wide. The average water depth is about 5 m, with a maximum depth of about 7 m.

The River Yssel is connected to the lake by the Ketelmeer, a channel about 10 km long and 1 to 4 km wide, see figure 7.3. It runs at an angle of approximately 45 degrees with the longitudinal axis of the lake. The size and orientation of this channel make it difficult to represent it realistically in the computational grid used, the size of which is determined on the basis of the size of the main lake (see below).

The bottom topography of Lake Yssel is shown in figure 7.3. The depths are with reference to the New Amsterdam Level (NAP). Except close to the lake boundaries,
Figure 7.3. Bottom topography of Lake Yssel,
— grid boundaries; — — actual boundaries. * grid point (9,14)
— 4.0 — isobaths with depth in m below NAP.
variations in the bottom level across the lake area are moderate. The bottom slope is typically less than 1:1000. Because of old tidal channels the isolines of bottom elevation as a whole run in the longitudinal direction, with a shallowing towards the west and east banks.

The isolines shown were obtained through interpolation of observed bottom elevations with an accuracy of about 10 cm.

_inflows and outflows, free surface elevation_

Lake Yssel receives inflow from the River Yssel. Lake water is discharged into the North Sea through two sluices at the dam at the north end of the lake. Sea water is prevented from entering the lake by the dam.

The River Yssel has an average discharge of about 300 m³/s, corresponding to a residence time of the lake of approximately one year. The discharge during the period of the field measurement is shown in figure 7.4.

Along the lake banks there are several water intake points for agricultural purposes. However, the involved quantities of water are quite small compared to the discharge of the River Yssel.

The average free surface level is regulated using the two sluices located at Den Oever and Kornwerderzand (figure 7.1). On the average the total discharge of the two sluices is approximately equal to the discharge of the River Yssel, although it is influenced by tidal
variations in the Wadden Sea. Because of the difference between inflow and outflow, the free surface elevation of the lake is not constant throughout the year. It can vary from -0.5 m NAP to +0.10 m NAP.

**Wind conditions**

Records of wind velocities at three meteorological stations of the KNMI (Dutch Royal Meteorological Institute) near Lake Yssel were available for the period of the field measurement. The locations of these stations are shown in figure 7.1. The observations at the three stations showed little differences, which indicates that the wind velocity distribution across the lake area was quite uniform. The available wind records are hourly averaged wind velocities, and have been corrected for the possible influence of obstacles near the measuring stations.

One can convert the wind velocities into wind shear stresses to be used as input in a numerical flow model. The following empirical formula can be used:

\[
W = c_d \rho_o |u_{10}| u_{10},
\]  

(7-1a)

where \( \rho_o \) = density of air; \( u_{10} \) = wind velocity measured 10 m above the free water

---

**Figure 7.5. Wind shear stresses.**

--- x-component; -- y-component;
surface; \( c_d = \) drag coefficient. The value of \( c_d \) has been estimated according to (Hicks, 1972)
\[
c_d = C_1 + C_2 \cdot |u_{10}|
\]
where for Lake Yssel \( C_1 \approx 0.72 \times 10^3 \), \( C_2 \approx 0.061 \times 10^{-3} \) s/m.

The wind shear stresses computed using the wind records and equations (7-1) were made available to the author by Rijkswaterstaat-RIZA. They are shown in figure 7.5. These are components of shear stresses with respect to the model axes (see figure 7.3). It is seen that in general, the wind was characterized by short bursts of strong wind (of the order of several hours to one day), superimposed on slower variations of longer period (several days). The variations in the wind shear stresses as a whole were quite marked.

Two periods of strong wind were observed, one from 2 till 8 October, the other from 25 till 30 October. The wind was relatively weak between these periods and before the first period.

7.2.3 The observations

Evolution of solute cloud

Part of the observed concentrations are shown in figures 7.12a through 7.17a. The isolines of concentration were obtained by using linear interpolation. It should be pointed out that the measurement points are quite few and sparse. It is questionable at some instances as to whether the cloud surfaces are totally covered by these points. The patterns of the solute cloud shown by the interpolated isolines can therefore be quite artificial (also see Discussion of this section).

It is difficult to obtain from these figures a complete insight into the evolution process of the entire solute cloud. It seems that the solute cloud first migrated in the western direction close to the south bank (figure 7.13a). It made a turn at the south-west corner, then continued its way up towards the north bank of the lake (figures 7.15a to 7.17a). During this process of migration the cloud continued to grow. Areas with higher concentrations were found at the fore-front of the progressing cloud, and towards the west bank of the lake (see
figures 7.14a to 7.17a).

Upon examining these figures in further detail, one may find evidence of the presence of another branch of solute cloud that slowly moves towards the north bank alongside the east bank of the lake. Two separated patches of solute cloud are seen in figure 7.14a. Besides an area of higher concentration in the south-west corner of the lake, a similar area is seen off the east bank near Urk. Examining the subsequent measurements on 11 October (not shown), however, the latter patch of solute mixture seemed to have been lost. In the following measurements on 15 October (figure 7.15a), a long stretch of solute cloud, clearly separated from the western branch, appeared again along the east bank. One may therefore have missed this patch in the previous survey. Further evidence of the existence of an eastern branch of solute cloud is found in the measurement on 22 October (not shown) and 25 October (figures 7.16a). In the measurements on 22 October only one single patch of solute mixture with higher concentrations is seen near the west bank, at the height of Enkhuizen (the western branch). In figure 7.16a, however, an area of higher concentrations can be seen near the east bank, at about the height of Lemmer. This patch, clearly separated from a central cloud that is most probably part of the western branch, could be the eastern branch perceivable in figures 7.14a and 7.15a. The solute was spread over a large part of the lake area on 1 November, and it is difficult to find any distinct features in the concentration distribution (figure 7.17a).

**Maximum concentration**

The maximum measured concentrations in Lake Yssel are shown in figure 7.6. The size of the solute cloud was small and the concentrations were high shortly after its appearance in the lake. The peak concentration recorded was highest on 1 October, about 105 μg/m³. It then decreased rapidly during one week, and more gradually thereafter.

One should note that, however, these values are only indicative. Firstly, the route of measurement on each day did not cover the whole area of the solute cloud, which was unknown a priori. Secondly, the number of measurement points for one day of observations is quite small. Patches of solute cloud with peak concentrations might have been missed by the observers. In general, one would expect higher actual peak concentrations.
Discussion

The concentration distributions shown in figures 7.12a to 7.17a are obtained from sample measurements by using linear interpolation. They should be considered as first approximations, since the sampling points are quite sparse, especially at a later stage of dispersion when the solute cloud became larger (see figures 7.13a through 7.17a). This could particularly affect the accuracy of observed concentration distributions in areas of concentration peaks. Also, each survey took 6 to 7 hours on the average (see section 7.2.1). The evolution of the solute cloud during this time interval, which could be significant, is not taken into account.

7.3 Simulation of flow in Lake Yssel

The hydrodynamics of Lake Yssel are studied in this section using the quasi-3D numerical flow model DUTRID. Definitions of the computational grid and various model parameters are given first. Details of the velocity distribution in the lake are then examined for constant winds. Afterwards, computations simulating the flow under real wind conditions during the experiment are presented.
7.3.1 Computational grid

In the numerical simulations Lake Yssel was represented by a horizontal grid of $20 \times 35$ points in the horizontal plane (see figure 7.3). The grid size was 2 km. The available bottom elevation data correspond to a grid spacing equal to 1 km. They were averaged for the grid spacing used.

It is seen that on the whole, the grid represents the horizontal geometry rather adequately in spite of the large grid size. However, this grid is not fine enough to represent the Ketelmeer, the channel connecting the River Yssel to the main lake. The simulation of the flow in this channel is therefore only approximate. Inaccuracy was found to be introduced in the simulated transport time of solute through the channel. As will be shown in the next section, this may influence the simulation of the dispersion process of solute in the lake.

A finer computational grid was not considered in this study because of the limited computer resources.

Five grid points in the vertical direction were used to compute velocity distributions. Although this number seems small, it should not considerably influence the accuracy of predicted concentrations, as was shown by simulations of 2D dispersion in a channel with similar depths, presented in chapter 6. Note that this number is equal for all verticals so that the vertical grid spacing is proportional to the local water depth.

7.3.2 Model parameters

Several parameters in the numerical flow model must be defined for simulating the flow in Lake Yssel. The most important ones are the bottom roughness height and the horizontal eddy viscosity, which are introduced below. The algebraic vertical eddy viscosity model obtained in chapter 4 was used in the simulations. The Coriolis parameter is equal to $1.146 \times 10^{-4} \text{ s}^{-1}$ for Lake Yssel. The time-step used was equal to 20 min, corresponding to a Courant number equal to about 5.

**Bottom roughness**

Bed resistance to flow depends upon the physical roughness of the bed and the local
flow conditions near the bed. In the quasi-3D model DUTRID the bottom friction is modeled by using the law of the wall and a roughness height \( z_0 \), which is proportional to the representative height of the bed roughness element \( k \):

\[
z_0 = \frac{k}{30}.
\]  

(7-2)

The representative roughness height \( k \) is determined by the configuration and physical composition of the bed. The bed of Lake Yssel is quite smooth, composed of fine sand and mud. Two values were therefore considered: \( k = 0.06 \) m (a reasonable value for a bed with sediment transport) or \( k = 0.006 \) m (for smooth bed). Constant values were assumed for the representative roughness height over the whole flow field.

The bottom shear stress can also be expressed as a function of the depth-averaged flow velocity and the Chézy coefficient \( C \) or the friction coefficient \( F_r \) (see section 3.4). This is the formulation commonly used in 2DH flow models. For open channel flow over a hydraulically rough bed the friction coefficient is related to \( k \) according to (Rouse, 1938)

\[
\frac{1}{\sqrt{F_r}} = 5.75 \log \left( 12 \frac{D}{k} \right). 
\]  

(7-3a)

The Chézy coefficient is related to this coefficient by

\[
C = \sqrt{\frac{g}{F_r}}. 
\]  

(7-3b)

Both formulations of the bottom shear stresses will be used in the numerical simulations of flow in Lake Yssel presented below, and the results are compared. Note that in the flow model DUTRID, coupling between the 2DH module and the velocity profile module through the bottom shear stress is optional (see section 3.6.1). When uncoupled, the 2DH formulation of the bottom shear stresses (equations (7-3)) is used.

*Horizontal eddy diffusivity*

The exchanges of mass and momentum in the horizontal direction are commonly modelled using a horizontal eddy diffusivity \( E \). Knowledge about this diffusivity is
incomplete at present, and it is usually determined using empirical relations.

More understanding of horizontal exchanges can be gained by examining the structure of turbulence in the simulated flow. In shallow water flow the length scales of the turbulence range from a typical water depth to a characteristic length in the horizontal plane. This length is usually several orders larger than the water depth.

Turbulence with a length scale comparable to the water depth is of three-dimensional nature. Direct simulation of turbulence with length scales of the order of the water depth would therefore require a grid distance comparable to the vertical grid spacing. This part of the turbulence is modelled in DUTRID using an algebraic model (see chapter 5). Turbulent motions with length scales larger than the grid size in the horizontal direction can be reproduced by the flow model.

Turbulence with length scales equal to or smaller than the grid size and larger than the water depth, is not directly simulated. These turbulent motions are related to the large-scale variations of velocities in the horizontal direction, and are usually called the Sub-Grid-Scale (SGS) turbulence. A sub-grid-scale eddy diffusivity $E^{SGS}$ representing dispersion by the SGS turbulence is then introduced. The horizontal eddy diffusivity therefore consists of two parts:

$$ E = E^{3D} + E^{SGS} \quad (7-4) $$

where $E^{3D}$ is related to three-dimensional turbulence. For bed-generated turbulence, a rough estimation of $E^{3D}$ is $0.1 u^* D$ (Rodi, 1980; see also chapter 5).

Little is known about SGS turbulence till now. One can make a rough estimation using a Smagorinsky eddy diffusivity model (Smagorinsky et al, 1965). Assuming a local equilibrium of production and dissipation of turbulent energy in the horizontal plane, we have

$$ E^{SGS} = (\zeta \Delta)^2 \sqrt{S} \quad (7-5) $$

where $\zeta$ is a constant ($\approx 0.1$, Deardorff, 1970); $\Delta$ is the grid size in the horizontal direction; and $S$ is one-half of the squared shear rate, or:

$$ S = 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2. \quad (7-6) $$

It must be noted, however, that the Smagorinsky model does not take into account the influence of wind shear stresses and bed-friction on SGS turbulence.
For typical flows in Lake Yssel we can assume: \( u \sim v \sim 0.1 \text{ m/s}; u' \sim 0.01 \text{ m/s}; D \sim 5 \text{ m}, \) and

\[
\frac{\partial u}{\partial x} \sim \frac{\partial u}{\partial y} \sim \frac{\partial v}{\partial x} \sim \frac{\partial v}{\partial y} \sim \frac{0.1}{2000} \text{ 1/s},
\]

which gives \( E^{\text{ED}} \sim 0.005 \text{ m}^2/\text{s}, \) and using \( \Delta \sim 2000 \text{ m}, E^{\text{SGS}} \sim 5.7 \text{ m}^2/\text{s}. \) It is seen that in this case \( E^{\text{SGS}} \) is much larger than \( E^{\text{ED}}. \)

The values of the horizontal eddy diffusivity used in the numerical simulations were based on these estimations.

7.3.3 Circulations induced by constant wind

**Depth-averaged flow**

Numerical experiments were made to examine circulations induced by constant north and east wind, respectively. The wind shear stress was assumed equal to 0.4 N/m², approximately corresponding to the largest recorded wind velocity in the field experiment. Computations start from still water conditions and continue till a steady-state is reached. Computations were made with different values of the horizontal eddy viscosity \( E \) and the roughness height \( k \) (see below). Here only the results for \( E = 0.5 \text{ m}^2/\text{s} \) and \( k = 0.006 \text{ m} \) are shown. The number of grid points in the vertical direction used in the computations was equal to five.

One finds complex flow patterns in the computed circulations (see depth-averaged flow patterns in figures 7.7 and 7.8). A typical flow pattern known as topographic gyre is clearly seen in the computed circulations for the north wind (figure 7.7). This wind is approximately in the direction of the longitudinal axis of the lake. Note that in part of the lake the isolines of the bottom elevation tend to run parallel to this axis (see section 7.2.2). When the steady-state is reached a free-surface gradient in the down-wind direction is established. In shallower parts of the lake near the west and east banks, the force caused by the pressure gradient is small compared with that due to the wind shear stresses. In these parts the water is therefore transported in the wind direction. In deeper parts near the central axis of the lake, the force
Figure 7.7. Depth-averaged steady-state flow induced by north wind.
Figure 7.8. Depth-averaged steady-state flow induced by east wind.
caused by the pressure gradient is larger than that due to the wind shear stresses. Water in these parts therefore flows in the opposite direction. In this way, closed gyres are formed in the horizontal plane.

One can find similar structures in the velocity distributions driven by east wind, although in a less pronounced form (figure 7.8).

The differences between the flow patterns for the north and east winds are marked. There is a strong dependence of the velocity distribution in the lake upon the wind direction. Under practical conditions the direction of the wind changes continually (see figure 7.6). One can therefore expect to find a highly unsteady flow pattern for real situations.

Since the rates of the inflows and outflows are quite small, the influence of these flows upon the circulation in the lake is quite weak. Nevertheless, computed flows in the Ketelmeer for the north and east wind show little difference because the inflow is dominant in this part of the flow field.

Three-dimensional structure of flow

Wind-induced circulations in a closed water body exhibit a strong three-dimensional character. As was shown above, complex flow patterns appear in the horizontal plane under the combined action of wind and pressure gradient. A complex structure of the velocity distributions also exists in the vertical direction. In a typical water column, water near the free surface is driven by the dominant wind shear stress and is transported in the direction of the wind. A counter-flow in the reverse direction, driven by the pressure gradient, may exist in lower layers close to the bottom in part of the flow field with local water depths larger than, say, the averaged depth. As a result, the horizontal velocity vectors typically rotate over the water-depth.

The Coriolis force also causes the velocity vectors to rotate over the vertical.

Figures 7.9 shows the horizontal velocity distributions at three levels close to the free surface, to the bottom and near mid-depth, respectively, for the north wind. Note that these
Figure 7.9. Velocity distributions in Lake Yssel (north wind);
(a) level 1 (near the bed); (b) level 3; (c) level 5 (near the free surface).
levels are not horizontal because of the uneven bed. It is seen that variations in flow
directions over the water depth are quite marked. In the larger part of the flow field the
velocity vector rotates over the vertical. As a general tendency, flow velocities are
approximately in the wind direction in the water layer closest to the free surface (figure
7.9c), and in the reverse direction in the layer closest to the bottom (figure 7.9a).

**Influence of formulation of the bottom shear stresses**

Depth-averaged circulations can also be computed using a 2DH model. In such a
model, the bottom shear stresses are usually assumed to be functions of the depth-averaged
flow velocities, and the Chézy coefficient is used to represent the relative bottom roughness.
Physically, the bottom shear stress vectors are related to and in the direction of the local
velocities near the bottom. In wind-induced flow the local velocities near the bottom are at
large angles with the depth-averaged velocities in most cases (compare figures 7.9a and 7.7).
Large inaccuracies can therefore be introduced by using such a formulation of the bottom
shear stress.

Furthermore, there are uncertainties about the Chézy coefficient to be used with wind-
induced flows. For open channel flow, a simple relation exists between this coefficient and
the physical bottom roughness height (see equations (7-3)). In case of wind-induced
circulations in closed water bodies, however, this relation can no longer be applied. The
Chézy coefficient then is a function of the flow conditions, as is shown below.

Consider a simple case of two-dimensional flow in a vertical plane in a straight
horizontal channel. For a free surface gradient not equal to zero, the vertical distribution of
the velocities is given in equation (5-6) (see section 5.2.3). In deriving equation (5-6) a
parabolic distribution of the vertical eddy viscosity was assumed. The depth-averaged
velocity $\bar{u}$ can be obtained by integration:

$$
\bar{u} = \frac{gDi}{\kappa u^*} \left( (\xi - 1) \ln \left( \frac{D}{e_0} \right) + 1 \right) \quad (7-7)
$$

where $i$ denotes the free surface gradient and
\[ \xi = \frac{|W|}{\rho g Di}. \]

Assuming a depth-averaged flow velocity in the negative \( x \)-direction and using the Chézy coefficient, the bottom shear stress can be expressed as \( \rho g \bar{u}^2/C^2 \). Since the total horizontal momentum is equal to zero when the steady state is reached, we have:

\[ |W| = \rho g Di - \frac{\rho g}{C^2} \bar{u}^2 \]

from which equation

\[ \bar{u} = -\frac{C}{\sqrt{g}} \sqrt{1 - \xi} \sqrt{gDi}. \quad (7-8) \]

From equations (7-7) and (7-8) we have

\[ C = \frac{\sqrt{gDi}}{u^*} \frac{\sqrt{g}}{\kappa} \left( \sqrt{1 - \xi} \ln \left( \frac{D}{z_0} \right) - \frac{1}{\sqrt{1 - \xi}} \right). \quad (7-9) \]

For uniform open channel flow without wind action, we have \( \xi = 0 \) and \( u^* = \sqrt{gDi} \). Equation (7-9) then becomes

\[ C = \frac{\sqrt{g}}{\kappa} \left( \ln \left( \frac{D}{z_0} \right) - 1 \right). \]

This equation is similar to equations (7-3).

In wind-induced flow, \( C \) depends on \( \xi \), that is, the relative importance of the wind shear stress compared with the pressure gradient term. This dependence is stronger when \( \xi \) approaches unity, that is, when these two terms are equally important in the momentum equations.

For \( \xi \neq 0 \), a usually much larger value of \( C \) is obtained with equations (7-3) than with equation (7-9) for the same roughness height \( k \). Since the depth-averaged flow velocities are proportional to the Chézy coefficient, an overestimation of these velocities is made when using equations (7-3).

Test computations using DUTRID were made for a decoupled solution of the velocity distributions. The formulation of the bottom shear stress then is as in a 2DH model. The flow conditions were the same as in the computational example presented above, with a
Figure 7.10. Depth-averaged steady-state flow induced by north wind, computation using a Chézy formulation of the bottom shear stresses.
constant north wind. The value of the Chézy coefficient was determined using equations (7-3).

The resulting depth-averaged velocity distribution, shown in figure 7.10, is to be compared with the coupled solution shown in figure 7.7. It is seen that the 2DH model indeed considerably overestimates the depth-averaged velocities. The overestimation is especially large (up to 5 times) in areas of the flow field where the velocity is small. In these parts of the flow field the pressure gradient term and the wind shear stresses are nearly balanced (ξ approaches unity). In other parts of the flow field this overestimation is less (20 to 100%). Examining figures 7.10 and 7.7 in further detail, differences in the flow patterns, though small, can also be detected.

In summary, the formulation of the bottom shear stress using the depth-averaged velocities and the Chézy coefficient is inappropriate in wind driven flows. The Chézy coefficient then is a function of the flow conditions. With this coefficient determined according to equations (7-3), an overestimation of these velocities is made. A formulation based on the law of the wall and the local velocities near the bottom, as is used in DUTRID, is a physically more realistic approximation.

Influence of model parameters

The model parameters, especially the bottom roughness k and the horizontal eddy viscosity E, may also influence the computed velocity distributions. Numerical experiments were therefore made with different values of these two parameters: k = 0.06 m or 0.006 m; E = 0.5 m²/s or 5.0 m²/s. For all cases a constant north wind was assumed, with the wind shear stress equal to 0.4 N/m².

The computed velocity distributions are not shown here for reasons of brevity. The following conclusions were drawn from these test computations.

1. For the case with E = 0.5 m²/s or 5.0 m²/s, little difference was found in the magnitudes and distribution of the flow velocities;

2. Using the smaller roughness height (k = 0.006 m) yielded a general increase in the velocities of up to 10 ~ 20%, while the flow pattern remained unchanged.
7.3.4 Simulation of flow during the experiment

In section 7.3.3 circulations in lake Yssel induced by constant wind actions have been examined. Distinctive flow patterns can be observed in the simulated results, which strongly depend on the wind direction. Wind velocities recorded during the field experiment show marked and continual variations in both magnitude and direction. The largest wind velocities in periods of strong wind are about 15 m/s. The actual velocity distributions of flow in the

Figure 7.11. Time-histories of wind shear stress and the depth-averaged velocity at grid point (9,14). Figure above: \(-\cdots = W_x/\rho, \quad \cdots = W_y/\rho;\) figure below: \(-\cdots = \tilde{u}, \quad \cdots = \tilde{v}.\)
lake induced by the unsteady wind action during the experiment may have been quite complex.

Using the surface shear stresses derived from the wind records as input (see figure 7.11a), computations were made simulating the circulations in the lake during the experiment. The simulations covered a period of about 42 days, starting on 21 September. Still water was assumed as the initial condition. The wind shear stress was assumed to increase linearly from zero to the values derived from the first wind records within the first six hours. A time-step equal to 20 min was used in the simulations.

The velocity distributions obtained using DUTRID show a highly unsteady character. It is impossible to show all the results here. However, the time histories of the velocity components at grid point (9,14) are shown in figure 7.11b as an example. The position of this point in the lake is marked in figure 7.3. Variations in velocities because of long wave oscillations have much shorter periods (about 6 and 1.5 hours for oscillations in the longitudinal and transverse directions of the lake, respectively) than variations in wind velocities. These oscillations can be identified in the figure, especially for the velocity component \( \bar{v} \). Oscillations of \( \bar{u} \), the amplitude of which is much smaller than that of \( \bar{v} \), cannot be easily seen in the figure because of the low resolution of the plot. The difference in the amplitudes of oscillations of the two velocity components \( \bar{u} \) and \( \bar{v} \) is caused by the fact that these oscillations follow the transverse and longitudinal directions of the lake, respectively. One may also observe that the two components appear to be quite correlated, which indicates the existence of a local principal flow direction (east-west). Such a direction may exist because the local isobaths are also in this direction.

### 7.4 Simulation of dispersion of solute

As was mentioned in section 7.2.1, Rhodamine used in the field dispersion experiment is a passive, soluble chemical. The particle dispersion model presented in chapter 6, which runs in conjunction with DUTRID, can therefore be applied to simulate dispersion of this solute in Lake Yssel. In this section results of such simulations are presented and compared with the measurements. Influences of various aspects of the simulations upon the computed
results are also examined.

7.4.1 Particles representation of amount of solute

In the particle model the solute is represented by a finite number of discrete particles. The amount of solute represented by each particle must be determined. This can be done when the total amount of solute is known. In addition, discharge of solute should be converted into the number of particles per time-step of computation.

The discharge of solute of the River Yssel can be computed using the measurements of water discharge and concentration of solute in the river water. These measurements were made at Kampen (see section 7.2). Only advective transport was considered at the discharge point. As can be estimated from the concentration measurements (figure 7.2), the local concentration gradient is rather small. The dispersive transport, which is proportional to this gradient, is therefore neglected.

Using these measurements, the total amount of solute \( M \) discharged into Lake Yssel during the period considered can be determined. It was found to be equal to about 13.53 kg.

Suppose this amount of solute is represented by a total number of \( N \) particles, each representing an amount of solute \( M/N \). The number of particles to be discharged in one time-step of computation is then equal to the amount of solute being discharged within that time interval, multiplied by \( N/M \), then converted to an integer using truncation. The total number of particles actually discharged in all time intervals is counted (usually less than \( N \) because of the truncations), and is used to recalculate the mass one single particle actually represents. The vertical position of a particle in the water column at the moment of injection is determined at random, using a uniform probability density.

7.4.2 Simulation of dispersion of solute

Model conditions

Constant inflow and outflows were assumed in the numerical simulations. The discharge of River Yssel was taken equal to 300 m³/s. Slight modifications in the solute
concentration at the discharge point were made, such that the solute mass discharged within each time interval remains unchanged. Discharges at the two sluices (at Den Oever and Kornwerderzand, see figure 7.1) were assumed equal to 120 and 180 m³/s, respectively.

The average free surface elevation was assumed equal to -0.20 m NAP. The wind shear stresses were as given in figure 7.11a. Other assumptions made in the computations were:

- $E$ (horizontal exchange coefficient) = 10 m²/s. This value is somewhat higher than what was estimated in section 7.3.2. It gives a better fit between the computed concentrations and the measurements (see the sensitivity study presented in section 7.4.3). Assuming Reynolds analogy, this coefficient was used for exchanges of both mass and momentum;
- $z_0$ (bottom roughness parameter) = 0.006 m;
- A shorter length of the Ketelmeer (see section 7.4.3 for details);
- Number of grid points in the vertical = 5;
- Total number of particles $\approx$ 5000;
- Time-step of computation = 20 min.

**Concentration distributions**

Results of concentration distributions from the numerical simulations are shown in figures 7.12b through 7.17b. These are instant concentration distributions at 12 a.m. on each day indicated. These results can be compared with the measurements shown in figures 7.12a through 7.17a.

The computed overall process of evolution of the solute cloud in the lake compares fairly well with the measurements. Upon entering the lake, the solute cloud soon splits into two smaller patches. One of the two patches moves towards the west bank, changes direction at the south-east corner, then moves up towards the north bank while always remaining close to the west bank. The process of migration of this patch of solute cloud was well identified in the measurements (see section 7.2.3). The other patch of solute, only partly seen in the experimental results, seems to move all the way towards the north bank, while remaining close to the east bank. The transitional movements of the two patches are quite independent
Figure 7.12a. Concentration distribution on 1 October (in $\mu g/m^3$), measurement.

Figure 7.12b. Concentration distribution on 1 October, computation.
Figure 7.13a. Concentration distribution on 4 October (in $\mu$g/m$^3$), measurement.

Figure 7.13b. Concentration distribution on 4 October, computation.
Figure 7.14a. Concentration distribution on 8 October (in \( \mu g/m^3 \)), measurement.

Figure 7.14b. Concentration distribution on 8 October, computation.
Figure 7.15a. Concentration distribution on 15 October (in µg/m²), measurement.

Figure 7.15b. Concentration distribution on 15 October, computation.
Figure 7.16a. Concentration distribution on 25 October (in μg/m³), measurement.
Figure 7.16b. Concentration distribution on 25 October, computation.
Figure 7.17a. Concentration distribution on 1 November (in µg/m³), measurement.
Figure 7.17b. Concentration distribution on 1 November, computation.
until they more or less merge at a later stage (around 25 October, figure 7.16b). During this process of migration, the two patches spread in the transverse direction because of turbulent dispersion.

The computed values of the concentration are in fair agreement with the measurements, especially in areas traversed by the left branch of the solute cloud. The high concentration zones in the solute cloud approximately coincide with those found in the experiment.

Upon entering the lake the computed size of the solute cloud is large compared with that in the measurements. This can presumably be partly attributed to errors introduced by the computational grid used in the simulations. The representations of the Ketelmeer as well as the lake boundaries near the entrance of this channel in this grid are quite crude (see next section). Another source of error is possibly in the eddy diffusivity assumed. At the initial stage of mixing the size of the solute cloud could be small compared with the largest turbulence eddies in the flow. The actual diffusivity may therefore be smaller than that assumed in the computations.

The subsequent splitting of the solute cloud near the entrance of the Ketelmeer (figure 7.13b) has a strong influence upon the entire dispersion process. It can also be detected in the measurement (figure 7.13a), though much less pronounced than in the computed result. The splitting is caused by the presence of large eddies in the area above the south bank (see figures 7.7 and 7.8, for example), which interact with the flow induced by the discharge of the River Yssel.

The results of the computation for 8 October compare quite well with the measurements (figures 7.14a and 7.14b). A much less good agreement is seen for 15 October. The high concentration zone in the left branch is much closer to the west bank than in the measurements, and concentrations in the eastern branch are high.

The computed concentrations are in reasonable agreement with the measurements on 25 October. However, no structured pattern of the solute cloud can be discerned in the measurements. The measurement points were quite sparse.

Intense dispersion took place from 25 till 30 October because of strong wind. After this period, small patches of solute cloud with high concentration disappeared into the background concentration. By 1 November, the solute cloud occupied a large part of the lake surface (see
figure 7.17b).

The computed concentrations on 1 November are generally higher than those measured. A possible explanation is the known fact that Rhodamine is absorbed by mud. Part of the solute could have been lost into the bed with sinking mud particles so that it was no longer observed.

**Maximum concentration**

Maximum concentrations from the numerical simulation are shown and compared with the measurements in figure 7.6. As could be expected (see section 7.2.3, discussions), the computed results are higher (up to 20 to 60%) than observed, except in the first week after discharge of solute in the lake. Note that the computed concentrations are values averaged over a large area (4 km²). The actual maximum concentrations may be even higher.

**CPU time**

The numerical simulations covered a period of about 42 days, starting at 21 September. With a time-step equal to 20 min, a total number of about 3000 steps of computation was performed in each series of simulations. The simulations were made on a Hewlett-Packard work-station (series 9000/400).

One 42-days numerical simulation took about 30 min CPU time on the work-station. Most of the CPU time was consumed by the particle tracking procedure. The computation of the flow took only one sixth of the total CPU time.

**7.4.3 Sensitivity study**

**Dispersion of solute in the Ketelmeer**

The entire dispersion process of the solute in the lake depends upon the instant the cloud enters the lake, and the initial concentration distribution shortly after the solute cloud has entered the lake. This instant and the initial distribution obviously depend upon the flow
in the *Ketelmeer* as well as the flow in the lake near the entrance of the channel.

Modelling of dispersion in the *Ketelmeer* was necessary because water discharges were known for *Kampen* only.

Modelling of the flow in this part of the flow field was very approximate because of the small size of the channel compared with the grid size used. The width of the channel decreases on approaching its entrance into the lake, where it becomes as small as about half the grid size. The flow structure near the entrance cannot be properly modelled with the present grid, but could be important for the initial exchange of solute inside the lake (see figures 7.12a and 7.12b).

Modelling of flow and dispersion in the *Ketelmeer* could be refined using a smaller grid size. In this study, however, this possibility was not worked out.
Measurements of concentrations in the *Ketelmeer* at the entrance were available (see figure 7.2). Considerable differences are seen across the channel section, which shows that the solute was not yet fully mixed across the channel.

In the computational results the progression of the solute cloud in the *Ketelmeer* was slightly slower than in the measurements because the grid required a larger width (figure 7.3). It was therefore necessary to reduce the length of the channel (by removing three grid points marked by a cross in figure 7.3). Figure 7.18 shows the concentration distribution on 15 October computed without shortening the channel. It is seen that the western branch of the solute cloud begins to stretch itself in the longitudinal direction of the lake slightly later than in the result with the original length of the *Ketelmeer* (figure 7.15b).

![Computed concentration distribution on 15 October, influence of wind shear stress.](image-url)
Influence of parameterization of wind shear stresses

Wind shear stresses used in the numerical simulations presented in section 7.4.2, were computed using records of wind velocities and the empirical equations (7-1). In all the computations presented above it was assumed in equation (7-1b) that \( C_1 = 0.72 \times 10^4 \), \( C_2 = 0.061 \times 10^3 \) s/m.

The wind shear stress is the main driving force of circulation in the flow field. The influence of the parameters in equation (7-1b) on the computed concentration distributions therefore could be large. This was confirmed by computations made with different values of these coefficients (\( C_1 = 0.76 \times 10^3 \), \( C_2 = 0.079 \times 10^3 \) s/m). Figure 7.19 shows the computed concentration distribution on 15 October, which can be compared with the previous result shown in figure 7.15b. A large difference can be seen, particularly between the positions of the western patch.

Influence of the horizontal eddy diffusivity

A discussion about the horizontal eddy diffusivity used in the numerical simulations was given in section 7.3.2. Dispersion of solute in the horizontal plane is enhanced by both velocity shear in the vertical plane and turbulence. Because of the large grid distance used in the present simulations, the SGS turbulence dominates over the three-dimensional turbulence as far as the horizontal exchange is concerned.

Dispersion due to velocity shear in the vertical plane can be estimated according to Taylor (1954), see Fischer et al (1979). Assuming a logarithmic velocity distribution (applicable for open channel flow), the coefficient of shear dispersion (\( K \)) is approximately equal to \( 6.0 \, \nu^* D \) (see equation (6-29), section 6.5.1). For wind-induced flows the dispersion coefficient is larger, say, \( K = 12.0 \, \nu^* D \). For flows in Lake Yssel we can typically assume: \( \nu^* \sim 0.01 \) m/s; \( D \sim 5 \) m, in which case \( K \sim 0.6 \) m²/s. This coefficient can be compared with the eddy diffusivity accounting for the SGS turbulence. According to the estimation given in section 7.3.2, \( \varepsilon_{\text{SGS}} \sim 5.7 \) m²/s, which is much larger than \( K \).

The computational results presented in section 7.4.2 were obtained using a horizontal
eddy diffusivity equal to 10 m²/s. A simulation using a smaller eddy diffusivity (1.0 m²/s) was also made. The concentration distribution on 8 October from this simulation is shown in figure 7.20. Considerable differences are found between these results and those presented above (see figure 7.14b), although the cloud patterns are similar.

It is seen that the influence of the SGS turbulence upon dispersion of substances is large. This is a typical feature with shallow water modelling. Modelling of this part of the total turbulence in the present numerical models is purely empirical. One could consider reducing the SGS eddy diffusivity by using a smaller grid size. However, according to the estimation presented in section 7.3.2, this diffusivity decreases approximately linearly with the grid size. In order to substantially reduce the SGS turbulence, the total number of grid points would have to be drastically increased. A more practical approach would be the implementation of a better formulation of the SGS turbulence. This would be a subject for further research with the present flow and dispersion models.

Influence of the number of grid points in the vertical

The accuracy of prediction of the vertical distributions of the horizontal velocities depends on the number of grid points used in the vertical direction. As far as modelling of
of the solute is concerned, this has consequences upon the accuracy of the dispersion caused by velocity shear in vertical planes.

Five grid points were used in the vertical in the numerical simulation presented in the preceding section. A simulation using a larger number of grid points (10) was also made, with other parameters being unchanged. The concentration distribution on 8 October from this simulation is shown in figure 7.21.

Very little differences in the computed concentration distributions are found (compare figures 7.21 and 7.14b). Possible explanations for the good agreement are: firstly, modelling of velocity distributions in the vertical direction using the present flow model is quite accurate, even with a small number of grid points (see chapter 5). Improvement in the computed results with more grid points is therefore small. Secondly, since the total dispersion is dominated by advection and the SGS turbulent transport (see above), any improvement in the part of dispersion associated with vertical shear is hardly noticeable in the concentration distributions.
Influence of time-step

The accuracy obtained in modelling velocity distributions is influenced by the time-step used. A time-step equal to 20 min was used in the computations presented in section 7.4.2. In order to examine the consequence of the time-step upon the concentration distributions, computations were made with a time-step equal to 10 min. The results for 8 October are shown in figure 7.22. It is seen that they are virtually identical with those presented in section 7.4.2 (c.f. figure 7.14b).

Figure 7.22. Computed concentration distribution on 8 October, using a time-step equal to 10 min.

Influence of the number of particles used

In the numerical simulation presented in section 7.4.2, the total amount of solute was represented by 5,000 particles. Errors introduced by this approximation could be large. Computations were therefore made with nearly 10,000 particles. Computed concentration distributions were compared with those presented in section 7.4.2.

Little difference was found between the two sets of results, see, for example, the concentration distributions on 8 October shown in figures 7.23 and 7.14b for comparison.
Figure 7.23. Computed concentration distribution on 8 October, using 10,000 particles to represent the solute mass.

The size of the solute cloud increases in the process of dispersion. At the end of the simulated period the cloud occupied nearly 250 grid cells. With 5,000 particles representing the total amount of solute, an average number of about 20 particles is found within each grid cell. This number seemed large enough for the present simulations. Nevertheless, although the average error in the computed concentrations may be small, the errors are obviously larger at areas populated with few particles, that is, areas with small concentrations.

One should also note that the computed concentrations are values averaged over the area of one grid cell, which is quite large (4 km$^2$). A higher resolution may be obtained by averaging over a smaller area. However, errors introduced by factors such as the lower resolution of flow field and the approximation made in modelling the SGS turbulence, may also need to be reduced.

7.4.4 Comparison with 2D computations

It was shown in section 7.3.3 that a 2D depth-averaged flow model is inappropriate for simulating wind-induced flows. The flow velocities are usually overestimated with such a model, especially at locations where the wind shear stress is approximately equal to the
Figure 7.24. Concentration distribution on 8 October, computed using a 2DH flow model and the particle model.

pressure gradient term.

In order to examine how this effect influences the accuracy of the predicted concentration distributions when a 2DH flow model is used in a dispersion model, a numerical simulation of the field experiment was made in which the velocity distributions are computed using DUCHESS. Assuming a uniform velocity distribution in the vertical direction, the particle tracking model can then be applied without modification. The bottom friction coefficient in the 2DH model is determined using equations (7-3), assuming $k = 0.006$ m (the same as in the 3D computations) and $D = 5$ m (average water depth of the lake).

Two of the computed concentration distributions are shown in figures 7.24 and 7.25. Compared with the results computed with the Q3D flow model (see section 7.4.2), fairly large differences can be observed, especially at a later stage of the dispersion process (compare figure 7.25 with 7.17b). However, these differences seem to be less significant in terms of agreement with the measurements. At certain instances the agreement is slightly better for the 3D computations (compare figures 7.24, 7.14a and 7.14b), at other instances the agreement is to the same degree with both computations. It is noted that the comparison
Figure 7.25. Concentration distribution on 1 November, computed using a 2DH flow model and the particle model.
is made difficult by the lack of a sufficient number of measurement points available (compare figures 7.25, 7.17a and 7.17b, for example).

Still, observing the large discrepancy between the velocity distributions computed with the 2DH and the Q3D flow models, one would expect larger differences in the computed concentration distributions. A possible explanation for the results obtained is that the discrepancies in the velocity distributions are relatively small in regions of high velocities (see section 7.3.3). Since flow in these regions plays a more important role in the dispersion process, the resulting discrepancies in the concentration distributions are also small.

In order to examine the computational efficiency of the quasi-3D flow model DUTRID, simulations of the flow during the experiment were made using this model and a 2DH model (DUCHESS), respectively. Computations using DUTRID took about 5 min CPU time for one simulation. This is nearly 2.5 times that needed by the 2DH simulation. Another simulation with DUTRID using 10 grid points in the vertical took 3.3 times the CPU time needed with DUCHESS. The difference in the CPU times needed by both models is small when simulating the dispersion process. As was explained in section 7.4.2, the particle tracking procedure takes the largest part of the total CPU time.

### 7.5 Discussion and conclusions

A field experiment on dispersion of a solute (Rhodamine) in Lake Yssel was presented in this chapter. The quasi-3D flow model DUTRID and the particle model of dispersion of passive solute developed in the present study, were used to simulate this experiment. The computed concentration distributions of solute were compared with the measurements, and in general the agreement was satisfactory.

The hydrodynamics of the lake was studied using DUTRID in section 7.3. Most of the input variables to the numerical model, such as wind conditions, were determined using data provided by *Rijkswaterstaat-RIZA*. Complex flow patterns are seen in the flow field because of the bottom topography, even for steady and homogeneous wind action. The flow showed
a strong three-dimensional character, with marked variations in flow velocities over the water depth as well as in the horizontal directions. Temporal variations in wind conditions measured during the experiment were considerable, which resulted in even more complex velocity distributions that continually changed with time.

The presented simulations of circulations in Lake Yssel provide a test for the computational efficiency of the quasi-3D flow model DUTRID. A 3D simulation takes approximately 2.5 or 3.3 times the CPU time needed for a 2DH simulation, for a resolution of 5 or 10 grid points in the vertical direction, respectively. This shows that the quasi-3D flow model has a limited demand for computer capacities and is suitable for practical applications.

The dispersion of solute in the experiment was simulated using the particle model that runs in conjunction with the flow model. The results of concentration distributions are in fair agreement with the measurements. Test computations showed that the results markedly depend on the drag law used to calculate wind-induced surface shear stresses, and on the horizontal mass exchange coefficient, which was assumed constant in the numerical simulations. Dispersion resulting from SGS turbulence seemed important compared with dispersion due to velocity shear in vertical planes. Dependence of the results upon the bottom roughness, the time-step of computation, the number of grid points in the vertical direction and the number of particles representing the amount of solute in the particle model, on the other hand, was quite weak.

Numerical simulations of the field experiment were also made using a 2DH flow model and the particle model. Marked differences were found in the computed concentration distributions compared with those obtained using the Q3D flow model. The differences are less significant when comparing the simulations with the measurements.

The solute discharged from the River Yssel was first transported and mixed in a short channel (the Ketelmeer) before entering Lake Yssel. The size of the channel is rather small, which gave rise to difficulties with its presentation in the computational grid, the size of which is chosen on the basis of that of the main lake. Simulation of flow in the channel as well as that in the lake near the entrance was quite approximate. It was necessary to reduce
the length of the channel to obtain the right time of arrival of the solute cloud in the main lake.

The numerical simulations presented in this chapter can be refined in various ways. Firstly, a finer grid in the horizontal plane could be used, at least in sub-domains. Because of the large grid distance used in the present simulations (2 km), the lake boundaries were quite poorly represented for part of the flow field, especially with the Ketelmeer. This could have resulted in large inaccuracies in the predicted concentrations, since the solute cloud was transported close to these boundaries, at least at the first stage of dispersion (see figures 7.12 through 7.15). Secondly, further study could be made concerning the influences of various model variables, such as the SGS eddy diffusivity and the coefficients used to convert wind velocities into shear stresses. Computations showed that the concentration distributions noticeably depend upon these variables.
CHAPTER 8

SUMMARY AND CONCLUSIONS

*Quasi-3D numerical flow model DUTRID*

A quasi-3D numerical model of large-scale circulations in shallow water (DUTRID) has been developed. This model computes the full spatial distribution of velocities of unsteady flow induced by wind and tidal forces. It provides a useful tool in many applications involving hydrodynamics of shallow waters (e.g. lakes, estuaries and coastal zones), such as transport of sediment and dispersion of discharged pollutants.

The following assumptions were made in the numerical flow model:

* Shallow water assumption. The typical water depth of the water body is assumed much smaller than a typical length scale in the horizontal directions. Hence the hydrostatic pressure distribution approximation can be applied;

* Boussinesq assumption. The turbulent shear stresses are modelled by introducing eddy diffusivity coefficients;

* No vertical density stratification.

DUTRID has the following characteristics:

(1) Application of the Vertical-Horizontal Splitting algorithm. The velocity distributions in the horizontal plane and those in the vertical direction are computed separately in two modules. In the first module the depth-averaged shallow-water equations are solved; in the second module the vertical distribution of horizontal velocities is determined.

(2) Application of the law of the wall in modelling the bottom boundary condition. A
no-slip condition is assumed and the bottom roughness is represented by a characteristic roughness height. The bottom shear stresses are computed by using the nodal velocities nearest to the bottom.

The free surface boundary condition is approximated by equating the near-surface shear stress to the imposed wind shear stress.

(3) **Coupling of modules.** The depth-averaged module uses the bottom shear stresses computed in the velocity profile module which in its turn uses the free surface gradients that are computed in the first module.

As part of the coupling process, depth-averaged velocities computed in the depth-averaged module are used to make corrections to the computed velocity profiles. This correction is necessary because of conservation of mass and momentum.

(4) **Adective terms retained in both modules.** The advective terms are shown to be small for shallow water flow with a low Froude number. However, they may have noticeable consequences upon flows due to their cumulative effects, as was illustrated through a vorticity analysis. In the depth-averaged module these terms are treated according to the ADI algorithm. In the velocity profile module an explicit, first-order upstream scheme is used.

(5) **Stability.** The algorithm used in the depth-averaged module is unconditionally stable (N. Booij, 1989); the stability of the velocity profile module is limited by the Courant condition based on the convective velocity.

(6) **Accuracy.** High accuracy is obtained in solving the vertical distributions of velocities. Large gradients of flow velocities occur in the vicinity of the bottom and the free water surface exposed to wind shear. These gradients are well reproduced by the model, even with a small number of grid points in the vertical direction.

DUTRID was applied to simulate a number of flows with simple geometries, namely:

(1) **2D flows in straight channels** induced by a free-surface gradient, by surface wind shear or by a combination of both. Analytical solutions exist for the velocity distributions for a given distribution of the vertical eddy viscosity. The numerical results approximate the analytical solutions with sufficient accuracy.

(2) **Flow over a sloping bottom** (maximum bottom slope = 2%) in a straight channel. Marked influence of the advective terms on the computed vertical distributions of velocities
was found, despite the small bottom slope.

(3) Secondary currents induced by the Coriolis force in a straight channel with horizontal bottom. These secondary currents have the same characteristics as those induced by curvature of the main flow. Computational results of these currents were compared with the analytical solutions of Kalkwijk and Booij (1986). Good agreement was obtained.

(4) 3D circulations in a laboratory harbour basin opening to a tidal river. Typical features of these flows observed in a laboratory experiment (Langendoen and Kranenburg, 1990), such as secondary currents, were well reproduced in the computational results. A low resolution in the vertical direction was used in the computations (with 5 grid points). It was shown that these flow features originate from the advective transport of momentum.

Related to the development of the flow model, a study of the vertical eddy viscosity in flows driven by wind and a free surface gradient was made using laboratory measurements (Yu, 1987) and a k-\( \epsilon \) model. Various expressions for the distribution of the eddy viscosity were tested using DUTRID. The conventional parabolic distribution over the depth was found to give a remarkably good fit of velocity distributions to the measurements by Yu, provided the friction velocity is equated to the larger of the bed- and surface-friction velocities. Computations using a k-\( \epsilon \) model substantiated this formulation of the eddy viscosity. Wind waves were suppressed in the experiment. Their effects upon the turbulence are therefore excluded.

A particle model of dispersion of solute

A particle tracking model simulating dispersion of passive dissolved matter in turbulent shear flow in shallow water, has been devised. The model is based on the analogy between the Fokker-Planck equation of random walk and the advection-diffusion equation. The model runs in conjunction with the quasi-3D flow model DUTRID.

Compared with the classical method for solving the well-known advection-diffusion equation numerically on a Eulerian grid, a particle model typically has a number of advantages: no negative concentrations occur; numerical diffusion is absent and mass is
conserved automatically. However, a large number of particles may be needed to obtain an acceptable accuracy of the model results, which sometimes makes the model inefficient.

Advection is simulated by using a cell-analytical approach (personal communication, Stelling, 1991). The net drift of particles, typical of conventional numerical integration methods, is thereby reduced. This feature was illustrated for advection in a 2D rotating flow.

Diffusion in homogeneous and non-homogeneous turbulence is simulated by adding, at each time step, a random component to the advective displacement of particles.

The particle model was applied to shear dispersion in 2D shallow water flow in the vertical plane. Computations were made with various numbers of vertical grid points. The rate of increase of the variance of particle distribution in the flow direction was computed and compared with the analytical solutions given by Taylor (1954) and Elder (1959). Good agreement was obtained.

The influence of the maximum length of random displacements upon the accuracy of the results was investigated. Computations showed that this length can be equated to the local water depth rather than the vertical grid spacing. The resulting error in predicted concentrations was within acceptable limits for engineering applications. Since the time-step of particle tracking is proportional to the square of this length, the computational cost of the model is greatly reduced.

Simulation of a dye dispersal experiment in Lake Yssel

The flow and dispersion models developed in the present study were applied to simulate a dye dispersal experiment in Lake Yssel (surface area 1120 km²) in the Netherlands. This experiment was carried out in 1990 by Rijkswaterstaat-RIZA and the N.V. Watertransportmaatschappij Rijn-Kennemerland.

Numerical simulations covered a period of about 42 days during which the solute mass was dispersed over a large part of the lake. Constant inflows and outflows were assumed in the numerical simulations. The wind shear stresses were computed from wind records using an empirical relation. Constant values were assumed for the horizontal exchange coefficient
and bottom roughness height.

The hydrodynamics of the lake were studied assuming both idealized (constant) and real wind conditions. Complex flow patterns were seen in the horizontal plane because of the bottom topography and wind action. The flow showed a strong 3D character with marked variations of the flow velocities over the water depth.

This study also showed that the formulation of the bottom shear stresses using the depth-averaged velocities and a Chézy coefficient, is inappropriate for wind driven flows. Estimating the Chézy coefficient according to a conventional relation for open channel flow results in overestimation of the computed velocities.

This study also provided a test for the computational efficiency of the flow model DUTRID. A 3D simulation costs about 2.5 or 3.3 times the CPU time needed for a 2DH simulation, for a resolution of 5 or 10 grid points in the vertical direction, respectively. The computations were made on a Hewlett-Packard work-station (series 9000/400).

The numerical results for the evolution of the solute pattern and the concentration distributions are compared with the measurements. Fair agreement was obtained.

A sensitivity study showed that the computed concentration distributions markedly depend on the horizontal mass exchange coefficient and the wind shear stresses. The dispersion due to sub-grid-scale turbulence seemed to be important compared with that due to velocity shear in the vertical planes. On the other hand, varying the bottom roughness, the time-step of computation, the number of grid points in the vertical direction or the number of particles representing the solute mass in the particle model, seemed to have little influence on the results.

The representation of the small Ketelmeer in the relatively coarse finite-difference grid was quite approximate. Hence a large discrepancy resulted between the numerical results and measurements of the concentration distribution when the solute cloud just appeared in the lake after passing through the channel. In order to obtain the correct time of arrival of the solute cloud in the lake, it was necessary to adjust the length of the channel.

The numerical simulations presented in chapter 7 can be refined in various ways. A
finer grid in the horizontal plane, at least in sub-domains, could be used such that the boundary and topography of the lake are represented more accurately. Further study could be made concerning the influences of the sub-grid-scale eddy viscosity and the coefficients used to convert wind velocities into shear stresses.

The flow model DUTRID can easily be extended to take account of tidal variations of the free water surface and horizontal density gradients. The particle model can also be extended to simulate transport of materials other than passive solutes, such as sediments by introducing a fall velocity.
REFERENCES


CURRICULUM VITAE

The author was born in Hubei, China, on 20 July 1963. He went to Dalian (China) in 1979 and studied harbour engineering at the Faculty of Civil Engineering, Dalian University of Technology. He graduated with a Bachelor's degree in 1984. He was then sent to Dalian Foreign Language Institute for one year, learning Japanese, as preparation for going to Japan to continue studying harbour engineering.

Yet in February 1985 he found himself in Liège, Belgium. In the Université de Liège he learned French, then majored in coastal and offshore engineering in the Faculty of Civil Engineering. He obtained the degree of la Maîtrise en Sciences Appliquées in 1988.

In the same year, he decided to go to Delft University of Technology to continue his study for a Ph.D. degree. He joined, as a research assistant, the Hydromechanics Section of the Hydraulic and Geotechnical Engineering Division, Faculty of Civil Engineering of this university. He studied and made research on computational hydrodynamics, and completed this dissertation.

He joined Delft Hydraulics in October, 1992, where he is employed as a research engineer. He carries out research and project studies on transient flow in piping systems.