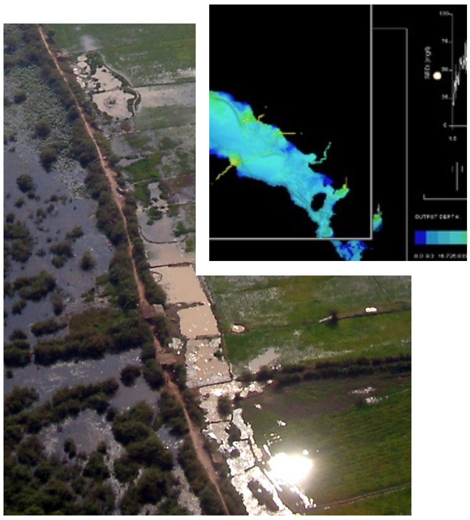
Mekong River Commission

Hydrological, Environmental and Socio-Economic Modelling Tools for the Lower Mekong Basin Impact Assessment



Water Utilisation Programme

WUP-FIN Phase2
Technical Paper No. 1

EIA 3D Model Manual

February 2008

Finnish Environment Institute
in association with
EIA Centre of Finland Ltd.
Helsinki University of Technology

Hydrological, Environmental and Socio-Economic Modelling Tools for the Lower Mekong Basin Impact Assessment

Water Utilisation Programme

WUP-FIN Phase II

Technical Paper No.1 – EIA 3D Model Manual

DRAFT

February 2008

Jorma Koponen, Matti Kummu, Hannu Lauri, Markku Virtanen, Arto Inkala, Juha Sarkkula, Ilona Suojanen, and Noora Veijalainen

Finnish Environment Institute
Mechelininkatu 34a
00260 Helsinki
Finland
Tel: +358-9-403 000
Fax: +358-9-40300 390
www.environment.fi/syke
juha.sarkkula@environment.fi

EIA Ltd.
Tekniikantie 21 B
02150 Espoo
Finland
Tel: +358-9-7001 8680
Fax: +358-9-7001 8682
www.eia.fi
jorma.koponen@eia.fi

Helsinki University of Technology
Water Resources Laboratory
Tietotie 1E
02150 Espoo, Finland
Tel: +358-9-451 3821
Fax: +358-9-451 3856
www.water.tkk.fi/wr
marko.keskinen@tkk.fi

This working paper is part of the WUP-FIN2 technical paper series. All the technical papers are listed below

Technical paper		Status
TP1	EIA 3D model manual	Draft
TP2	VMod Hydrological model manual	Draft
TP3	HBV Hydrological model manual	Draft
TP4	RLGis manual	Draft
TP5	Tonle Sap Lake water balance calculations	Draft
TP6	Database report	Draft
TP7	Remote sensing study of Mekong River bank location in Vientiane – Nong Khai area	Draft
TP8	Sediment dynamics of Tonle Sap Lake	Draft

This document can be referred as follows:

MRC/WUP-FIN, 2008. Technical Paper No.1 – EIA 3D model manual. WUP-FIN Phase II – Hydrological, Environmental and Socio-Economic Modelling Tools for the Lower Mekong Basin Impact Assessment. Mekong River Commission and Finnish Environment Institute Consultancy Consortium, Vientiane, Lao PDR. 291 pp. Available on-line at http://www.eia.fi/wup-fin/wup-fin2/publications.htm

OVERVIEW

EIA MODEL SYSTEM

The EIA 3D model system is fully three-dimensional (3D) model based on rectangular grid representation. The model system accommodates meteorological, hydrological, topographic, land use and infrastructure characteristics of any modelling area and produces 3D hydrodynamics and water quality. The modelling platform including data processing, model control, GIS, database control, model data products and visualization is de-coupled from the actual model engines. The model is able to describe the 3D characteristics of the flooding, flow, water quality, erosion and sedimentation in the lakes, reservoirs, river channels and floodplains.

The EIA 3D model is developed by Environmental Impact Assessment Centre of Finland Ltd (EIA Ltd.). The development work started 1974 when EIA Ltd. was still part of Technical Research Centre of Finland, the largest governmental research institute in Scandinavia.

Name of software:	EIA 3D Model
Developer:	Jorma Koponen, Markku Virtanen, Hannu Lauri, and Arto Inkala (of about 20 other developers) Environmental Impact Assessment Centre of Finland Ltd. (EIA Ltd) Tekniikantie 21 B, 02150 Espoo, Finland Tel. +358-9-70018680 Fax. +358-9-70018682 E-mail: koponen@eia.fi / virtanen@eia.fi
First available:	first 2D version 1975, first 3D version 1983, last major revision in 2002
Hardware required:	can be run in all types of computers from PCs to supercomputers
Software required:	can be run on all platforms (most user friendly version requires Windows 2000 or XP)
Program language:	FORTRAN (model and basic graphics) and C++ (graphical user interface)
Program size:	Hydrodynamics and other physical modules 1 Mb, Water quality and ecological modules 1.6 Mb, Graphical user interface 0.7 Mb (sizes are for Windows NT)
Availability:	Available free within cooperative projects or as part of an application
More information:	www.eia.fi

EIA 3D model can be classified as three-dimensional baroclinic multilayer model (Simons, 1980; Virtanen et al., 1986; Koponen et al., 1992). The water mass is treated as horizontal layers. Horizontally the model area is subdivided into rectangles with arbitrary mesh intervals in both directions. EIA 3D hydrodynamic model is based on the standard Navier-Stokes equations (1) in a rectangular grid. The cell width can vary in x- and y-directions. It is possible to model whole domains with varying grid resolutions and couple them together.

The model is solved numerically using implicit finite difference method applied to control volumes. For computational purposes the calculation of the 3D currents is divided into integrated 2D external mode (surface heights, depth integrated currents) and to 1D internal mode (layer velocity differences). Eddy viscosity approximation of turbulence is used with constant coefficients, also mixing length and k-e epsilon turbulence models are available. The advection of momentum has only minor effects on flows, when the flow velocities are small, and is therefore not always used.

WHAT'S IN THIS MANUAL?

The manual has been divided to four parts:

Part 0: Introduction and getting started
Part I: Model operation
Part II: Model description
Part III: Appendices

The structure of the manual and brief introduction for each chapter is provided below.

PART 0 – Introduction and getting started

1 Getting started

Chapter provides an overview for the EIA 3D model including e.g. its architecture, model engines, input and output data, model grid definitions, and main computational methods. The basic use of the model system is briefly introduced following by the introductions how this manual is built and can be best used.

PART I – Model operation

2 Condensed instructions for model setup and use

Short introduction how to set up and run a model simulation in different types of computation.

3 Model software

Chapter provides step by step instructions how to install the model software to your own computer following by the structure of the model software and model applications files and folders. Also the standard configuration of the model is presented.

4 Basics for using EIA 3D model system

In this chapter the basics for use of EIA 3D model system are presented including starting the model, graphical user interface, structure of the main menu and tool-bar. The opening and saving of existing model application are also described.

5 Creating new model application

This chapter introduces how to create a new model application either from elevation data or manually by creating empty grid.

6 View options

This chapter deals with the view options of the model graphical user interface.

7 Source data

This chapter deals with the source data of the model, its handling and importing. Different source data types are introduced.

8 Model Grid settings

This chapter deals with the model grid and settings related to it. It goes through the setting up of grid parameters and how the grid depths can be edited in the GUI.

9 Model variables

This chapter describes the model variables, including physical, landuse, water quality, and computational parameters.

10 Output

This chapter describes the model output options and settings including start and end state, and animation and timeseries options.

11 Fetch and dynamic fields

This chapter provides information for how to save fetch and dynamic output information, and then how to read and use the saved data.

12 Running the model

This chapter describes how the model computation can be started and model computation time set up.

13 Model Output Analysis tools

This chapter includes the instructions how to use the analysis tool for the field-type of results.

14 Animation

This chapter includes the instructions how to see the stored animation and use the animation tool.

15 Timeseries output

This chapter includes the instructions how to present the timeseries data and use the timeseries analysis tool.

16 Using help

This chapter includes the instructions how to use the help in the model.

17 Window management

This chapter describes the window management tools.

18 Troubleshooting

This chapter describes the most common trouble shootings in the model running.

19 Examples of model set-up

PART II – Model

20 Basic model equations

21 Mathematical Description of Water Flow

PART III – Appendices

TABLE OF CONTENTS

Table of Contents

ACRONYMS AND ABBREVIATIONS

2D	two-dimensional
3D	three-dimensional
BIL	Band Interleaved by Line, type of grid based GIS layer
BOD	Biological (biochemical) Oxygen Demand
COD	Chemical Oxygen Demand
EIA	Environmental Impact Assessment
EIA Ltd.	Environmental Impact Assessment Centre of Finland (www.eia.fi)
GIS	Geographical Information System
GUI	Graphical User Interface
HBV	Name of the lumped hydrological model
LU	Land Use
MRC	Mekong River Commission (www.mrcmekong.org)
OpenGL	Open Graphics Language / Open Graphics Library
PC	Personal Computer
RLGis	River Life GIS, a programme used together with the EIA 3D model
SOD	Sediment Oxygen Demand
SQL	Structured Query Language
VMod	Name of the 2D distributed hydrological model developed by EIA Ltd.
WQ	Water Quality
WUP-FIN	Lower Mekong Modelling Project under Water Utilization Programme of Mekong River Commission (www.eia.fi/wup-fin)

PART 0 – INTRODUCTION AND GETTING STARTED

1 GETTING STARTED

This chapter provides an overall introduction to the EIA 3D hydrodynamic model, the model's architecture and development. The chapter also gives a brief overview on the structure of the manual.

The chapter is divided to four parts:

1.1 Introduction to EIA 3D Model system and its development

The EIA 3D model system is fully three-dimensional model based on rectangular grid representation. The model system accommodates meteorological, hydrological, topographic, land use and infrastructure characteristics of any modelling area and produces 3D hydrodynamics and water quality. The modelling platform including data processing, model control, GIS, database control, model data products and visualization is de-coupled from the actual model engines. The model is able to describe the 3-dimensional characteristics of the flooding, flow, water quality, erosion and sedimentation in the lakes, reservoirs, river channels and floodplains.

The EIA 3D model is developed by Environmental Impact Assessment Centre of Finland Ltd (EIA Ltd.). The development work started 1974 when EIA Ltd. was still part of Technical Research Centre of Finland, the largest governmental research institute in Scandinavia. The EIA 3D model has two components: EIA 3D hydrodynamic model and EIA 3D water quality model. This manual concentrates on the first one but includes part of the water quality model as well.

The overview for the model developers and requirements is given in Box 1.

Box 1. Overview for the model developers and requirements.

Name of software:	EIA 3D Model
Developer:	Jorma Koponen, Markku Virtanen, Hannu Lauri, and Arto Inkala (of about 20 other developers) Environmental Impact Assessment Centre of Finland Ltd. (EIA Ltd.) Tekniikantie 21 B, 02150 Espoo, Finland Tel. +358-9-70018680 Fax. +358-9-70018682 E-mail: koponen@eia.fi / virtanen@eia.fi
First available:	first 2D version 1975, first 3D version 1983, last major revision in 2002
Hardware required:	can be run in all types of computers from PCs to supercomputers
Software required:	can be run on all platforms (most user friendly version requires Windows 2000 or XP)
Program language:	FORTRAN (model and basic graphics) and C++ (graphical user interface)
Program size:	Hydrodynamics and other physical modules 1 Mb, Water quality and ecological modules 1.6 Mb, Graphical user interface 0.7 Mb (sizes are for Windows NT)
Availability:	Available free within cooperative projects or as part of an application

Development work has required about 100 man years and three decades. Model system has been used in more than 250 research, engineering and consultancy studies.

1.1.1 Background of the model and work

The 3D lake and floodplain model is a new achievement based on over 100 man years of development and application work. Model development and application started in Finland in the 1970's in VTT (Technical Research Centre of Finland, the largest research institute in Scandinavia). Two of the main scientists (Koponen and Virtanen) are continuing this work up to today. Different 2D models including finite element methods were tested during the early period. Also some quasi 3D models were tested with reasonable success but emphasis on shallow water systems kept the main applications at first in 2D.

In the beginning of the 1980's development of 3D models started in earnest and since the 1980's main part of all hydrodynamic and water quality applications have been 3-dimensional. So far over 250 3D model applications have been conducted including harbour, bridge and road construction; reservoir modelling; waste water flow; cooling water intake for nuclear power plants; cooling water outlets; accident cases (oil- and chemical combating, sea rescue); waste water flow; toxic algae transport; erosion and flooding.

The early development work necessitated computational efficiency because of the limited computational power of the early computers. The efficiency has been one of the characteristics of the model code and solution algorithms until today. Other significant feature has been the use of field measurement data in all of the model applications for testing the model results. This has resulted in thoroughly verified solution methods. The long development history has eliminated practically all error sources from the code although of course every new development creates need to test the code.

In many cases the flow modelling has been combined with a specialized 3D water quality and ecological modelling. Solution method in the water quality model is optimized for speed and handling of complex data-sets. Calculated variables include nutrients, carbon, sulphur, silicate, oxygen, BOD, COD, suspended sediments, colour, turbidity, pesticides, radionuclides, organic chlorines and heavy metals by following processes: transport, diffusion, sinking, decay, sedimentation and leaching from the sediment.

Biological sub-models calculate phytoplankton, filamentous algae and dissolved nutrient (PO4, NO3 and NH4) cycles and optionally different size groups of bacteria, zoo- and phytoplankton. Bottom sediment processes and storages are modelled in three layers. Temperature, oxygen and carbon content affect the chemical processes. A specialized 3D reservoir model has been developed over many years for long term (decades) calculation of hydrodynamics and water quality in water bodies with high water level variations. Model has been specialized for speed and water volume and water quality constituent budget calculations.

Extensive work has been dedicated to distributed and physical hydrological modelling. Hydrological models are often combined with hydrodynamic and water quality models for rivers and lakes.

The model base and related graphics, data processing, GIS and user interface software consists of over 500’000 lines of code. About half of this consists of the 3D hydrodynamic and water quality model code.

1.2 Overview of EIA 3D hydrodynamic model

1.2.1 Overall design principles

The model system has been designed with a number of general principles:

- Data management and modification with an integrated GIS platform
- Common structural engine where application modules can be added
  - The application modules can be combined
  - The combined application modules will solve the actual modelling problem
- Data management and user interfaces are separated from the engine
  - Engine can be changed
- The ultimate design principles
  - Speed, calculation accuracy, robustness and error free code

The speed, robustness and error free code can in part be explained with the long development history and the limited computational resources that were available in the start of the development work in 1970’ies and 1980’ies.

1.2.2 Overall system architecture

Figure 1 shows the overall system architecture. At the basis of the system are three general databases: GIS, monitoring and meteorological. Data is processed to a specific model database that includes typically model grids and other input data in a format that the model engine can read directly. It should be observed that the original data is not intended to be modified by the modellers. Conversion happens either on the fly when the system is used or during the construction phase of the model application.

Model engines take care of grid construction, input- and output, data management, basic mathematical operations etc. Engines read the data from the model database. Various application modules are built on top of the basic engines. The application modules include river hydrodynamics, 3D hydrodynamics, hydraulic structures, water quality, erosion and hazardous materials.

Users modify the model data and model parameters, and control the simulation runs through an universal graphical user interface (GUI). The outputs of the model runs are used either locally or distributed into remote databases and users for further analysis.

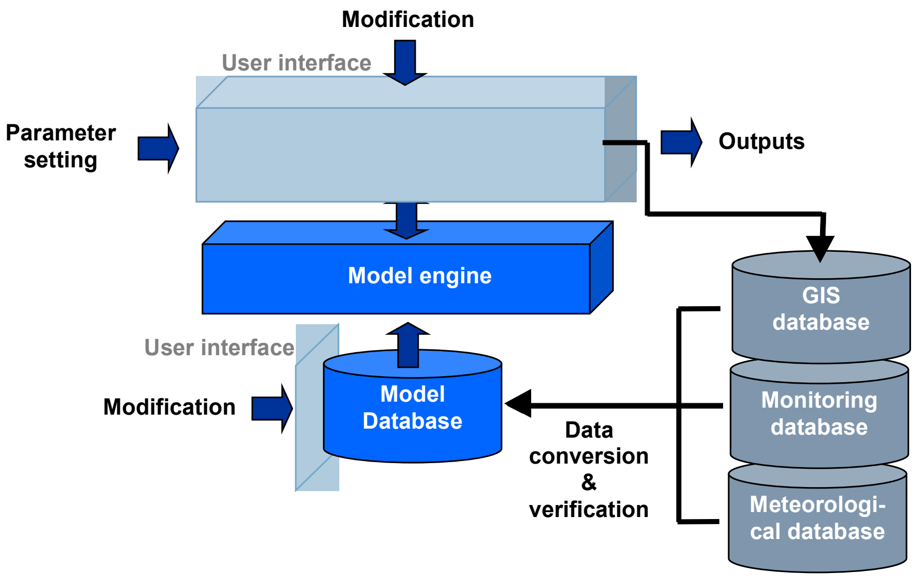


Figure 1. The EIA modelling system architecture.

1.2.3 Description of system components

The model engines consist of:

- two hydrological catchment models (HBV & VMod).
- various hydrodynamic models (EIA Hd)
- emergency, water quality, ecosystem etc. models (EIA Wq),

The lumped HBV hydrological model and VMod distributed watershed model provide data (water amounts, nutrients and sediments) for river, lake and floodplain models for flow, inundation, water quality, sediment, erosion and eutrophication.

Hydrodynamic models are divided into:

- 3D lake-, sea- and coastal (includes tidal shores)
- 3D reservoir
- 3D flood (rivers, flood wave propagation, dike and dam brakes)
- 3D diffusion wave (flooding, can be connected with 3D flood model)
- 2D Hansen
- Different 1D river models
- Hydraulic (e.g. gates, dike overflow in connection with other models)

Several models can be coupled to the hydrodynamics / hydrological models such as

- chemical processes e.g. evaporation, dissolution, emulsification on surface, in the water column and on bottom
- oil and chemical accident models, drifting objects (particle description in 3D)
- several water quality and ecosystem models for oxygen, BOD, turbidity, nutrients, heavy metals, carbon, different phytoplankton groups, macrophytes, bacteria etc.
- benthic processes

A separate geographic and time series data handling program, called RLGis, is included in the modelling system. RLGis is used to import, manage and prepare GIS and time series data for models, for example, to prepare a catchment model grid from a digital elevation model, and to import time series data from a SQL database.

In the following chapters the models and RLGis system are shortly described. To better understand the functioning of each of the models short descriptions of calculation principles included. For more detailed descriptions reader can look at the numerical model documentation in the appendixes and user interface help files.

1.2.4 General structure of the graphical user interface

The general structure of the user interface is shown in Figure 2. In all applications it is not necessary to realize all features.

For management and planning purposes the system can be run through the graphical GIS-user interface, where the user can define the calculation scenarios and viewing of the outputs. For the modelling specialists more comprehensive control of the model is necessary for setting up the model parameters, calibration, detailed model control and output analysis. The interface for the modelling system is general. In other words the interface is not coupled with any one model but can be used to generate input data for any model with sufficiently open input standards.

The specific features of the user interface are:

1. Integrated pre- and post-processing (model result comparison with measurements, change of bathymetry, statistics, presentation graphics etc.)
2. Graphics includes time-series, distributions and animations
3. Support for several graphics standards (3D OpenGL, Windows Metafile, PostScript, HPGL, Tektronix, raster formats)
4. Interface to GIS data (input and output in GIS-formats)
5. GIS-functionality in the user interface (overlays, analysis, map-based controls)
6. Remote running of the models over Internet
7. Direct link to SQL-databases
8. Data management tools for model input- and output data (e.g. initial values, inflows, weather, concentrations, time series output points)

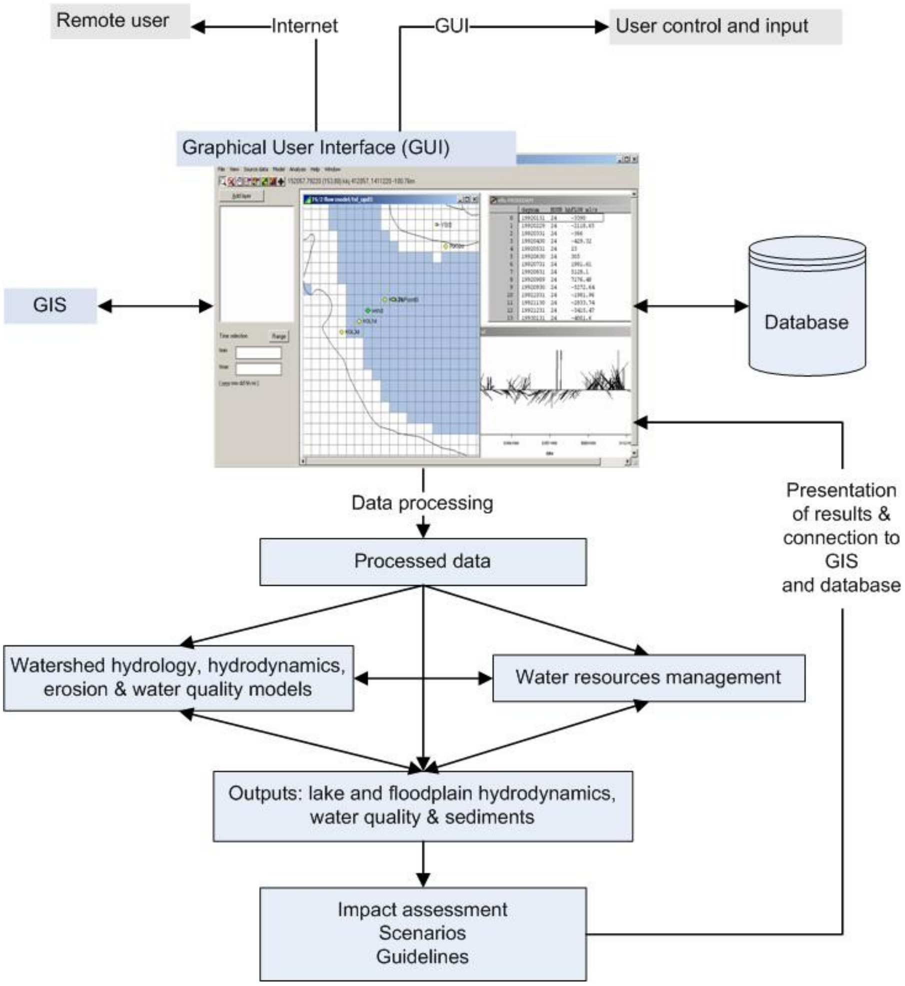


Figure 2. General structure of the system interface.

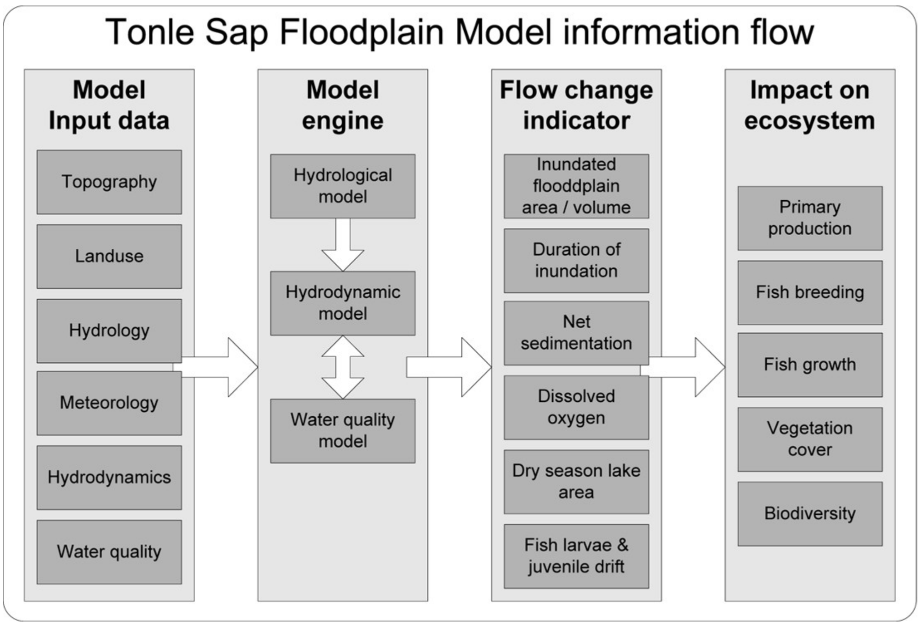


Figure 3. Model flow chart from model input to impact analysis.

1.2.5 Model applying concept

The general concept flow chart of applying the EIA 3D model system is presented in Figure 4.

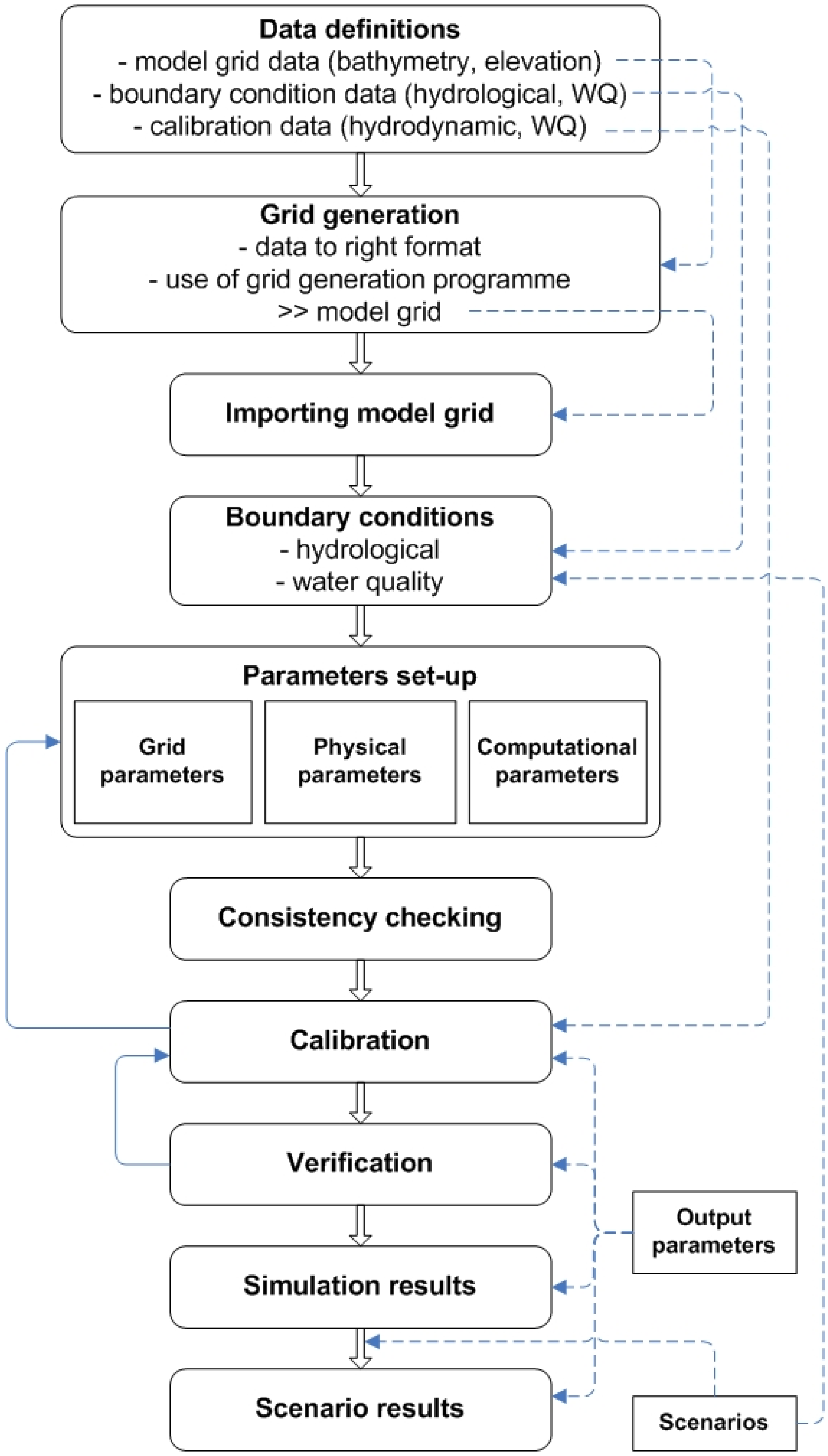


Figure 4. Flow chart of concept of applying EIA 3D model system.

1.2.6 The model data

The model data consists of temporal and spatial data. Data processing tools convert spatial data into model grids which are optimised for calculation accuracy and speed.

Model system can calculate efficiently large geographical areas by utilizing dynamically coupled model grids with varying accuracy.

Modelling system can read GIS data directly. Standard GIS tools can be used for data validation, checking and analysis. It is to be noted that the model construction process including specific model data processing and actual model calculation processes are separate from the rest of the system , e.g. from the GIS platform. This facilitates flexible use of alternative or complementary models when needed.

Monitoring data (historical & on-line) is directly accessed from the monitoring database. Because model results are often utilized extensively in GIS, all numerical model results can be output directly into a GIS database. EIA modelling system also supports large number of numerical and graphical formats. Examples of the latter include PostScript-, HPGL-, Windows metafile (clipboard) and Tektronix-formats. System creates accurate and space saving vector animations that can be viewed independently from of the modelling platform. Map data can be included in both animations and still pictures.

EIA modelling system includes model database management. The model metadata and actual data used in any model run (topography, parameters, source data file names, output specifications, comments, calculation results, graphics, animations etc.) can be viewed, edited, saved and deleted. These definitions can be used also for batch runs where a number of cases are required to be run one after another.

The design principles of EIA modelling system have been computational power, accuracy, adaptability to very wide range of problems, easy of use and quality of the technical implementation. EIA models can be used in PCs, workstations and mainframes under various operating systems. Code supports parallelization and vectorisation.

1.2.7 Model engines

Model engines provide a standardized set of data processing tools and mathematical solvers for physical and bio-geo-chemical processes. The main engines of EIA 3D model are:

- 3D hydrodynamic model (short summary in Section 1.2.8 - Hydrodynamic model computation method and in detail description in Chapter 22- The Numerical Flow Model)
- transport and water quality model (detail description in Chapter 23 - Transport and Dispersion Model)

Here only the first one is described in details. A separate manual exists for the transport and water quality model. However, the 3D hydrodynamic model described in this manual includes the options to calculate transportation of natural water quality parameters such as oxygen, sediment and temperature.

In practice the calculation of water currents is detached from that of material transport and water quality (Koponen et al., 1992) in order to save computation time. Short time-steps (often 10–30 s) needed in the calculation of flow velocities are unnecessary to repeat in the calibration runs of the water quality model.

EIA 3D model structure is presented in Figure 5.

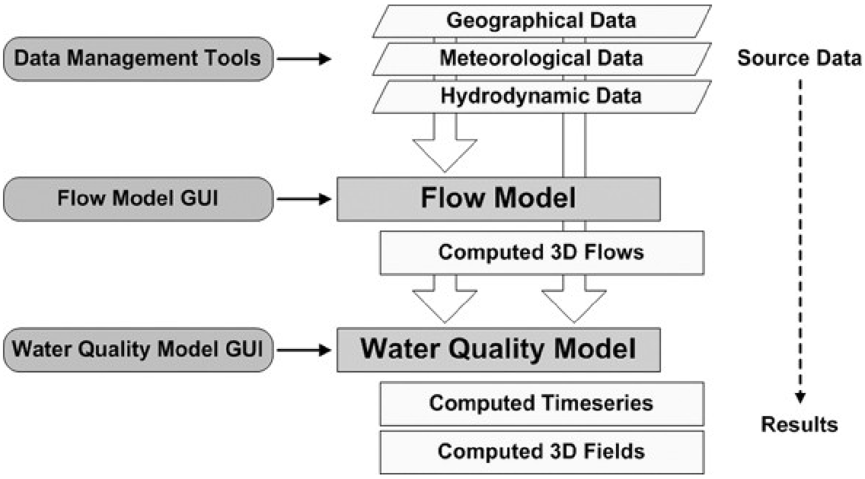


Figure 5. EIA 3D model structure.

The EIA 3D model used in the project is extremely versatile platform for very wide scope of applications. Some of the characteristics of the 3D model are:

- 6 vertical turbulence models (e.g. *k*-ε)
- 5 horizontal turbulence models (e.g. Smagorinsky)
- 2 integrated wave models (others in specialized applications)
- 2 wind fetch models
- 3 erosion models
- 4 bottom friction models
- Vegetation friction in different water layers
- Surface friction (e.g. ice)
- Radiation and heath
- Hydraulic controls (dikes, gates, water intakes, outlet points etc.)
- Ice formation and melting
- Wetting and drying
- Morphological changes due to sedimentation and erosion
- Mud layer simulation capability
- Specialized 3D reservoir model
- Diagnostic calculation from irregular data
- 2 isopycnal modes for stratification
- Hybrid stratification calculation (combined normal and isopycnal modes)
- 6 momentum advection modes (e.g. TVD)
- 3 transport calculation modes (e.g. TVD and flux correction)
- Integrated statistical analysis
- Algorithmic and code optimization resulting in fast execution times
- Parallelisation for multi-processor machines
- Flexible, fully coupled nesting for better local accuracy
- Transportable code (tested from supercomputers to PC’s)
- Code developed and tested over 20 years in over 200 applications
- Several models can be coupled to the hydrodynamics (hydrological models such as conceptual and distributed gridded watershed models; chemical processes e.g. evaporation, dissolution, emulsification on surface, in the water column and on bottom; several water quality and ecosystem models for oxygen, BOD, turbidity, nutrients, heavy metals, carbon, different phytoplankton groups, macrophytes, bacteria etc.; benthic processes)

The EIA 3D model can be used for wide range of applications:

- flood modules for dam- and dike breaks,
- detention area simulation,
- flooding of floodplains;
- reservoir models for managing reservoir hydrodynamics,
- sedimentation and water quality;
- water resources management;
- oil- and chemical accidents;
- monitoring support;
- and anoxia, eutrophication, algal blooms, filamentous algae, shore vegetation.

1.2.8 Hydrodynamic model computation method

EIA 3D model can be classified as three-dimensional baroclinic multilayer model (Simons, 1980; Virtanen et al., 1986; Koponen et al., 1992) and is based on solving simplified Navier Stokes equations (Equation i) in rectangular model grid. The cell width can vary in x- and y-directions. It is possible to model whole domains with varying grid resolutions and couple them together. Hydrostatic assumption, Boussinesq approximation and incompressibility of water are used in the model formulation. The water mass is treated as vertical layers similarly to z-level models. Horizontally the model area is subdivided into rectangles with arbitrary mesh intervals in both directions.

$$\tilde{\rho}\frac{d\vec{\tilde{v}}}{dt} = \tilde{\rho}\frac{\partial \vec{\tilde{v}}}{\partial t} + \tilde{\rho}\vec{\tilde{v}} \circ \nabla \vec{\tilde{v}} = -\nabla \tilde{p} + \tilde{\rho}\vec{g} \circ \vec{\vec{I}} - 2\tilde{\rho}\vec{\omega} \times \vec{\tilde{v}} + \tilde{\rho}\upsilon^{m}\nabla^{2}\vec{\tilde{v}} \qquad \text{(i)}$$

Where

$\vec{\tilde{v}}$	momentaneous flow velocity vector (m s^{-1})
$\tilde{\rho}$	momentaneous density of water (kg m^{-3})
$\tilde{p}$	momentaneous pressure (N m^{-2})
$\vec{g}$	gravity acceleration vector (m s^{-2})
$\vec{\vec{I}}$	unit matrix of the co-ordinate system (-)
$\vec{\omega}$	angular velocity vector of earth's rotation (s^{-1})
υ^{m}	kinematic, molecular viscosity of the water (m^2 s^{-1})
t	time (s)
∇	gradient operator (grad) (m^{-1})
$\nabla \cdot$	divergence operator (div) (m^{-1})
∇^{2}	Laplace operator (div grad) (m^{-2})

Explicit finite difference schemes are used for the numerical solution of flow velocities and water level elevations. The currents in the model are determined by the following factors:

- wind force (or ice friction),
- atmospheric pressure at the surface,
- conservation and incompressibility of water,
- internal friction (viscosity),
- transport of velocity differences with water currents (advection),
- Coriolis force,
- density differences and water level gradients (hydrostatic pressure),

- bottom friction
- vegetation impact

The model is solved numerically using implicit finite difference method applied to control volumes. For computational purposes the calculation of the 3D currents is divided into integrated 2D external mode (surface heights, depth integrated currents) and to 1D internal mode (layer velocity differences). Eddy viscosity approximation of turbulence is used with constant coefficients, also mixing length and k-e epsilon turbulence models are available. The advection of momentum has only minor effects on flows, when the flow velocities are small, and is therefore not always used.

1.2.9 Model grid

The model grid (see Figure 6) is based on the depths measured from the modelled area. Horizontal grid resolution depends on the application requirements, typical grid box sizes are from 50m in the area of interest up to tens of kilometres for large sea areas. Vertical resolution typically ranges from 0.5m on the surface to tens of meters in deeper areas.

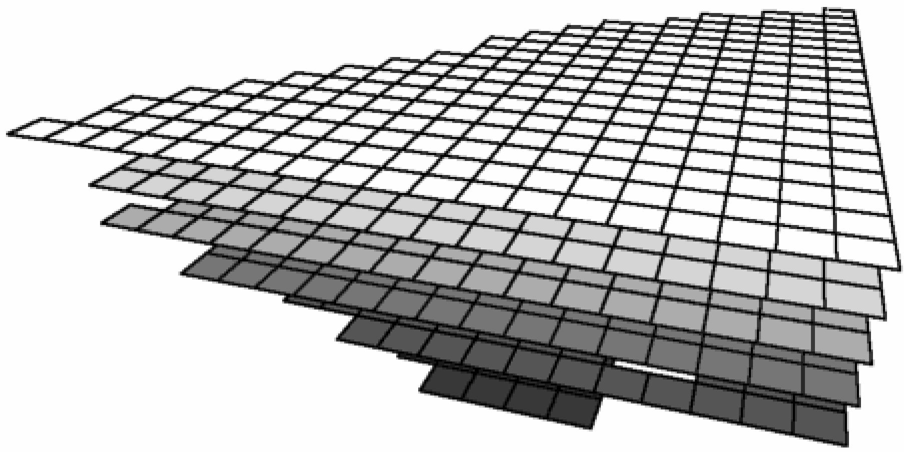


Figure 6. A side view of a simple 3d model grid – deeper vertical layers are shown in darker colours.

In the horizontal direction model utilizes rectangular Arakawa E-grid. In Arakawa E-grid both velocity components are defined in the middle of the grid cells as opposed to the usual C-grid arrangement where the components are defined on separate grid boundaries. In E-grid the vertical velocities are defined on the corners of the velocity grid cells. E-grid avoids stagnation points in 3D applications because water can circulate in the cells even if they are surrounded by land on each side. Model uses finite volume method to solve equations. Because of this the grid width can vary in x-, y- and z-directions.

Other common choices for grid system are orthogonal curvilinear and triangular meshes. Triangular meshes are used in FEM (Finite Element Method), but also in unstructured finite volume methods. Often the justification given for the use of these grid systems is that they can follow the river boundaries better. But also finite volume method can follow closely boundaries and a more important consideration is the model resolution that can be achieved with each method. Curvilinear coordinate systems create additional terms in the equations which decrease model efficiency. Also grid generation can be quite time consuming and the resolution cannot be focused freely on defined areas. The advantage of triangular grids is the very flexible selection of resolution. For instance the resolution can be easily focused on the river channel. The disadvantages of triangular meshes are at lessened computational inefficiency (FEM and algorithmic complexity in unstructured grids), more complicated software and more difficult coupling between GIS and model data. The numerical disadvantage of FEM is that mass conservation is only guaranteed with sufficient grid refinement whereas finite volume method always conserves mass.

In the vertical direction z-grid is used. This means that the layer depth remains constant over the whole model area except on the bottom where it varies freely. Because the Arakawa E-grid stagnation points are avoided and there is no need to utilize of coordinate system which has varying layer depths but constant number of layers in each grid point. s -system is computationally not as efficient as z-grid because latter has usually much less calculation points. Usually it is always advantageous to keep the vertical grid resolution constant over the calculation area because vertical properties are resolved in a consistent way over the whole model domain.

In the EIA model it is possible to couple different models with different resolutions together. In this way a large area can be modelled with very high resolution for critical areas. The nested models are fully coupled, in other words the high resolution model affects the coarse one.

1.2.10 Input and output data

Floodplain and lake modelling specific inputs are:

- topography (DEM)
- vegetation (roughness and friction)
- inflows and outflows
- inflow concentrations
- possible loadings
- winds

Grid generation programs read GIS-generated DEMs or other elevation data and produce the model grid. GIS can be used also directly to create the model grid but it has not yet been realized in the Tonle Sap system.

The data is read separately for each nested grid with different resolution. Based on the tributaries data model calculates the average width and depth of the tributary in each model grid cell.

Model outputs that can be used in the hydrodynamic part of the system include

- water depth (DEPS, DEPZ)
- water elevation (SURF)
- flow velocity components (U, V)
- flood duration (FLDU-files)
- flood arrival time (FLAR-files).

Other output parameters are e.g. concentrations, bottom heights especially in morphological studies, vertically averaged velocities, viscosity and other parameters connected to turbulence.

Model output files for GIS contain coordinate system and format specifications. The names of the files signify the output date and time, variable, layer and nested model. Model output resolution is user defined and is usually higher than calculation resolution. Interpolation and tributaries masking is used in the output. Output files are read directly into the GIS and are used e.g. for damage analysis and evacuation planning.

The GIS-format selected for the data exchange between the GIS and modelling software is BIL (Band Interleaved by Line). This format is open so that it can be accessed directly from the modelling software and is relatively efficient in storing data.

The main BIL-data files are accompanied by auxiliary files that define the binary format and coordinate system of the data.

Below in Figure 7 the flow model related information is shown. The model input data consists of model grid and model forcing data, e.g. wind and boundary flows. Output is 3-dimensional time-dependent flow field, which can be further used to compute, for example, transport of substances, water exchange, and sedimentation processes. Visualization of the flow data can be done with animations, and information of computed variables from single sites can be obtained as time series.

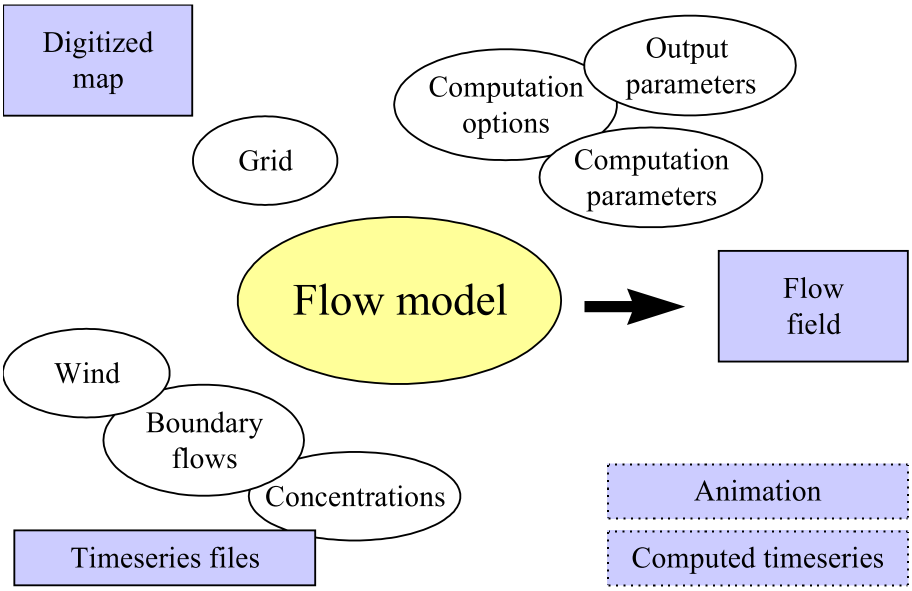


Figure 7. Flow model related information.

Input data summary:

- bathymetric data for model grid, either as shorelines, point depth data and depth isolines, or as a digital elevation model.
- wind measurements from modelled area for wind forcing computation, wind speed (m/s) and direction (degrees) with 3-6h or better time resolution
- boundary flows (m^3/s) including rivers and open boundaries (daily values)
- flow and/or water level measurements for model calibration, flow is often measured in cm/s for every ten minutes, for surface height the time resolution depends on the modelled area and may vary from 10 minutes to one day.
- temperature and salinity initial and boundary values (if computed).
- sources, initial and boundary values of transported substances (if computed).

Computed results summary (with any time resolution):

- computed 3d time-dependent flow field for the modelled area
- time series of flow speed, direction and water level
- time-dependent fields of other computed variables (temperature, salinity, suspended sediments etc.)
- time series computed variables
- animations of flow and computed variables

Model parameters

- The flow model parameters can be divided to several categories including:
- model setup parameters (calculation type selection related parameters)
- numerical parameters (computation time steps, etc.)

- physical parameters (e.g. turbulence coefficients, wind drag coefficient)

In addition to above model related parameters there are options for setting up the input and output data. The model parameters are explained more carefully in the model documentation and user interface help files.

1.2.11 Software aspects

Numerical and graphical calculation is realized with FORTRAN computer language. It is de-facto standard for numerical programming because its ease of use and computational efficiency. Interface to Windows calls is realized in C. Operating system dependent parts of the program are contained in specific modules and can be switched off by compiler directives.

Parallelization is achieved by general in-house parallelization software. The core routine for each process to be parallelized must be copied into specific parallel subroutine, but the management of modules is done with general routines. ADI and flooding/drying processes have not yet been implemented in parallel code.

Because model software development started in a period with modes computer power the code and solution algorithms have been designed to be efficient. Recently this requirement has been relaxed to some extent in favour of code maintainability because the code size has grown and the speed of computers has increased. Code speed is obtained in many cases by liberal use of memory, but so far standard or lightly above standard memory capacity has been enough. Nowadays the memory requirement is 256 MB – 512 MB depending on the grid size. The grid size reaches typically up to 1’000’000 grid points.

1.3 Using the flow model

There are several types of computations the flow model can be configured to perform including:

- computation of static flow field using constant external forcing (used to generate flow fields for water quality model),
- computation of dynamic flow using varying external forcing (used for model verification or generating flow data for water quality model),
- computation of advection of substance from known initial state,
- computation of spreading of substance from a non-moving source point(s).

1.3.1 Setting up a model grid and basic parameters

The first task in setting up a model simulation is to prepare and import a model grid. The model grid preparation can be done using the RLGis program or separate grid construction program. After the grid is ready it is saved to a file and imported to the flow model. A grid can also be imported from existing model application. After the grid is imported some basic model parameters must be set, including:

- model grid depth levels
- model computation time steps
- model variables
- wind and flow boundary data
- add time series points
- set time series and animation output parameters

1.3.2 Computation of static flow field

A static flow field is computed from an arbitrary initial state (usually zero flow), keeping all the external forcing constant, and computing the flow until is converges to a stable pattern. This is typically the simplest computation type since there is no need to set up time-varying external forcing and discharges.

Static flow field computation is useful to check that all the computational options are correctly configured and that the model computation does not get unstable after a long period of computation. Static flow fields can also be used in water quality model.

Typical static flow computations include situations where wind is constant, or there is a constant flow through the model area, or a combination of these. Note that a single constant flow coming into or leaving from the model area cannot be regarded as a static situation, since the amount of water in the model area either decreases or increases.

1.3.3 Computation of a dynamic flow field

Computation of time-varying flow differs from static flow field computation in such way, that in the static computation the external forcing are constant, whereas in the dynamic computation the forcing varies with time. Dynamic flow computation can be used, for example, to compare computed model results to measured flows.

1.3.4 Computation of advection of a substance from initial state

Computation of advection of substances from initial state is usually performed using a dynamic flow field. This kind of computation can be used to find out, for example, where a specific water mass is moving during a given time period. Also the exchange of water between different areas in different conditions can be explored.

1.3.5 Computation of spreading of a substance from a source point

Computation of spreading is usually performed using a dynamic flow field and initial values for the spreading variables. This kind of computation can be used to follow the spreading of one or more substances from one or more source sites.

1.4 Structure and use of the manual

This manual consists of three parts:

- **Model operation**: the main model operation functions are described in details with practical examples
- **Model description**: the model equations and theory are described in details
- **Appendices**: the model application examples are given among other things

The manual has been constructed and structured as user-friendly as possible to serve the needs of users with different backgrounds and purposes of model use. The manual includes illustrative figures and diagrams to make it easier for the user to follow and apply the information. The linkages between model operation and model description have been established to facilitate the better understanding of the theory behind the different parts of the model.

1.4.1 Structure of chapter and information boxes

Each **chapter begins** with the small text box with short introduction to the chapter and list of the sub-headings in the chapter including links to the places in text where the sub-headings are. The example of the small white text is provided boxes presented below:

> Text box at the beginning of each chapter works as a chapter introduction. Each sub-heading, as shown below) is linked to the place in question in text. Thus, it is easy to move to the point interested
>
> 0.1 [link to the sub-heading]

The steps to guide user through use of certain part of the model are described with the numbered lists:

1. the main steps are described with numbers:
    a. the sub-steps are described with letter

The **text boxes** provide more detailed information for example the parameters and other functions. The boxes also provides model examples based on the projects the model has been applied.

Box 2. Example of text box.

Aim of text boxes:	to provide more detailed and in depth information or theory background of the parameters, to offer modelling examples, etc

The **links** between Model operation and Model description parts are marked as an underlined blue text following the page number inside the brackets [page_number] where the link is pointing to.

This is the example of the link: See Box 1 [18]

The information box is marked with the blue info-symbol as illustrated below. The information box provides important information of the programme.

> The text box with the blue info symbol provides important information of the programme which should be taken into account when running the software.

The trouble-shooting information box is marked with the orange no-symbol as illustrated below. The information box provides information of the trouble shooting when possible malfunction of the programme occurs.

> The text box with the orange no-symbol provides information of the trouble shooting when possible malfunction of the programme occurs

The additional information box is marked with the blue computer symbol as illustrated below. This box provides e.g. more detailed information of the parameter, description of some term, etc.

The text box with the blue computer symbol provides additional information of the issue dealt in the text. This can be e.g. explanation of the parameter, description of some term, etc.

Whether you should have any further comments and/or questions related to the EIA 3D model system, this manual, or other matters related to the model or modelling work, please contact to EIA staff by email.

1.4.2 Conventions

This manual uses particular document conventions to help the user locate and identify information. The following typographical conventions are used in this manual:

Type style	*Used for*
Bold	Window or menu names available to be selected in EIA 3D model.
Bold	Fields that require the user to enter the required information such as, file names, data values or dates, etc.
<enter>	Requires the user to press the specified key
blue text	reference to other part of the manual or outside document

PART I – MODEL OPERATION

2 CONDENSED INSTRUCTIONS FOR MODEL SETUP AND USE

The general concept flow chart of setting up the EIA 3D model application is presented in Figure 8.

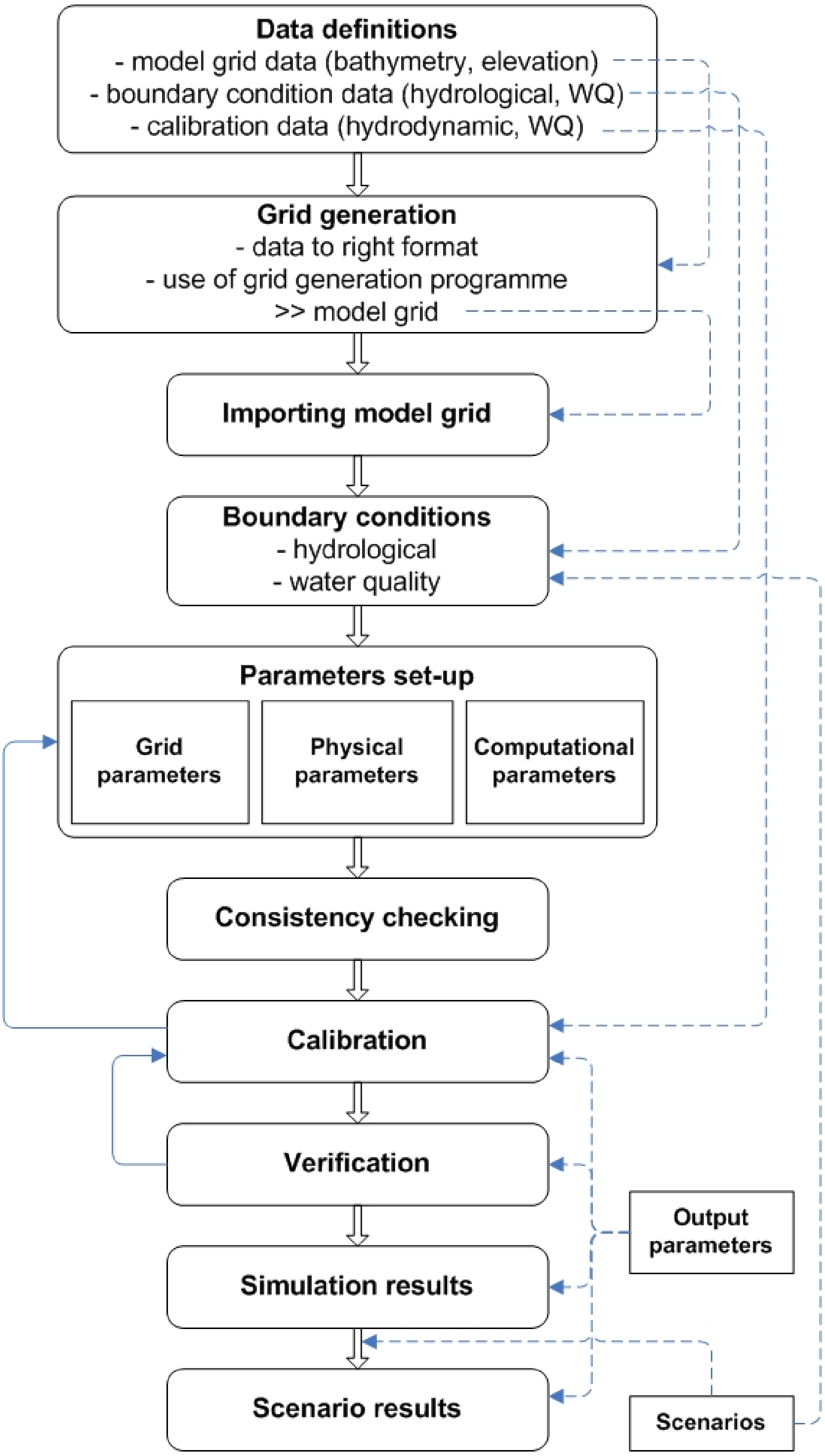


Figure 8. Flow chart of concept of applying EIA 3D model system.

The folder names in the instructions below refer to the MRCS Deliverables-package available through the MRCS/IKMP.

2.1 USER INTERFACE SOFTWARE INSTALLATION

To set up 3D model user interface, run **VivSetup.exe** in Deliverables\Software-folder. This should always be installed.

[*To run the distributed hydrological model (VMod) and the 1D channel model (RNet) one needs to change user interface version. By default VMod and RNet model user interface is installed into **c:\EIAModels\VIV-VMod** folder and 3D model user interface into **c:\EIAModels\VIV** folder. To change between the user interfaces/ folders use VivDirSetup.exe programme.*]

2.2 CREATION OF A NEW MODEL

Open any *.fld project-file in the ModelApplications sub-folders.

Click **File/New/Create empty grid**.

Specify file dimensions (x, y and grid size).

Click **Model/Grid depths** to specify the grid elevations (selecting tabular editing enables copying from outside sources). Observe that in the model – sign is inserted to the elevations to make them negative. This follows from the use of the model in two different modes either as (i) relative to a rather invariable water surface as positive depths or (ii) as elevations (negative "depths").

For storing a project click **File/Save**. For starting a new project select **File/Save as** and prescribe new name. Each time a project/model is run the previous output files including flow fields are overwritten if output file names are not changed either changing the names in the user interface or starting a new project with **Save as**.

2.3 SETTING UP A MODEL GRID

In case of a 3D application, set water layer depths (measured to the water surface) in **Model/Grid data/Edit level depths**.

Set up model grid location in **Model/Grid data/Grid coordinates.** Prescribe model grid origo coordinates (x,y) in the same reference system where maps are provided for the model and where model GIS outputs are used. Prescribe also model longitude and latitude. Only latitude has effect on the flow simulation through Coriolis-force.

2.4 SETTING UP COMPUTATIONAL OPTIONS

Select flow simulation, implicity and input modes:

Go to **Model/Computational parameters**.

Select **Calculation/ "external/internal"** for 2D + layer velocity difference mode (faster!).

Select **Calculation/layer** for direct calculation of the layer velocities (slower!, but in some cases such as high vegetation friction more accurate).

Check **predictor-corrector**-box for increased stability in high friction simulations (even then this is not often needed; makes simulation more slow).

For improved stability, check implicity for **Internal field**, **Vertical turbulence** and **Bottom friction** boxes. Instead of using implicity, other option is to use smaller time steps. **Internal field** implicity option requires that internal time step (external time step in case of the **layer** calculation mode) can be increased at least 100%. Otherwise simulation is slower than without the **Internal field**-option.

Select **Input/none**.

2.5 Setting up timesteps

Go to **Model/Time steps**.

Check **External field**, **Internal field**, **Bottom friction** and **Flooding** boxes and give appropriate time steps – as long as possible but preserving stability and physicality of the computation.

External field time step should be $\leq \Delta s / \sqrt{gH_{\max}}$, where Δs is the model grid size (m), g earth's gravitational acceleration (m/s^2) and H_{max} maximum water depth (m).

Internal field time step can be usually larger than the external one.

Flooding time step should be as large as possible (100 – 5'000 s) because flooding changes simulation topology and most of the model coefficients have to re-calculated making the flooding step computationally heavy.

Specify additional processes when needed (**Horizontal turbulence**, **Vertical turbulence** and **Nonlinear advection**, that is momentum advection). The additional processes can make the solution less stable requiring decreasing also other time steps.

2.6 Set up model parameter values

Go to **Model/Physical parameters**.

In **Vertical turbulence model** select either **Constant**, **k-e** (most universal) or **K+L**.

If **Constant** vertical turbulence is selected, provide viscosity values between layers by clicking **Constant vertical viscosity**.

Use small **Constant horizontal viscosity** value when the grid size is small (appropriate values for grid sizes less than 1 km about 1000 cm2/s).

Usually don't check **linear**-box in **Bottom friction**. Appropriate non-linear bottom friction values are 0.0025 – 0.01 depending on the flow and type of the bottom.

Set start-up **Water level** depending on your topographic reference system. For instance when the reference system is MSL (Mean Sea Level) in Hathien, the water level should be set around 165 m in Vientiane applications.

2.7 Set up animation and timeseries options

Go to **Model/Animation/Animation options**.

Select **Variable** from the list. For instance for flow speed select **SPED**.

Select appropriate **Scale**, for instance for **SPED** you may select 0 – 200 cm/s.

If flow arrows need to be drawn check **flow on**-box and define arrow **Coefficient**. The smaller the coefficient the smaller the animated arrows will be.

Prescribe **Animation frame step,** that is the interval how often animation is updated.

2.8 Set up flow control points and areas

Set up of boundary values, wind, load point, time series points etc. by selecting **Add item**-button from the control button row (yellow and green button with a +-sign). Click either one point or select an area with the mouse and select the control from the list opening by the mouse cursor.

2.9 Calculating, storing and using flow fields

Go to **Model/Start, end and dynamic fields**.

Check **Store end field**-box to store the fields at the end of the simulation and prescribe output file name.

For **Dynamic output** select output type. Usually select **aver. bin**, because it preserves flow volumes and is a compact binary format. Prescribe **Filename** and **Output timestep (h)**, that is the interval how often the fields are stored.

When using previously calculated flow field for start-up or sediment or water quality simulation, prescribe **Type** and **Filename**. For a dynamic binary average flow field the type should be **aver. bin** and for using end field of a previous simulation **ascii (EXV)**.

When using previously calculated dynamic flow field for prescribing flows for sediment or water quality calculation, go to **Model/Computational parameters/Input** and select type **aver. bin**. This option saves a lot of calculation time!

For storing illustrative statistics, data products or outputting GIS-files in BIL-format, go to **Model/Statistics**. Check **output 2D fields** and select parameters. Prescribe time interval for GIS-ouput when **output BIL files** is selected.

For running the model go to **Model/Run** and either prescribe run time in hours (**N hours**) or when **N hours** is zero or negative, **Start date** and **End date** for start and end of the simulation.

When running the model, animation can be updated more frequently than prescribed in the **Animations**-option by pressing **F3**-key when **OpenGl-animation window** is selected. For other options see OpenGl-menu in the animation window.

3 MODEL SOFTWARE

This chapter provides step by step instructions how to install the model software to your own computer following by the structure of the model software and model applications files and folders. Also the standard configuration of the model is presented.

The chapter is divided to three parts:

3.1 SOFTWARE INSTALLATION

Prior to use of the EIA 3D hydrodynamic model software the software has be installed to the computer. Follow the steps provided below in order to fulfil the setup procedure.

1. Double click the VivSetup.exe file

You have to be signed to the computer as an Administrator or user with administrator's rights to be able to install the software properly.

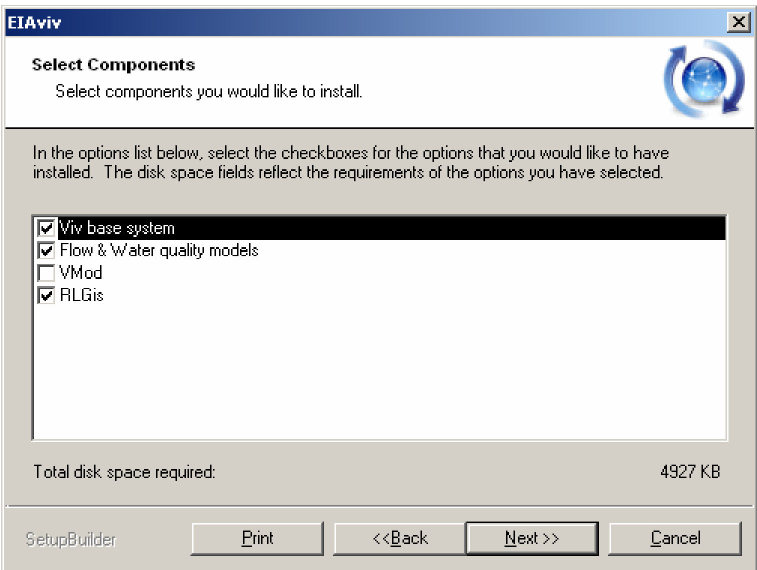
2. Press **Next >>** in the Welcome to setup window
3. **Software License Agreement:** Read carefully the "Master end-user license agreement" prior to accept the agreement by pressing **Yes**

4. **User information:** Write your name and company/organisation/university you are based in to the User information window. Accept the user information by pressing **Next >>**
5. **Select components:** Select the programme components you want to install. You have to select at least the following components to be able to run the EIA 3D model:
   a. Viv base system
   b. Flow & Water quality models

   Also the RLGis component is advised to be installed. RLGis is GIS programme supporting the files used in the EIA 3D model and needed for data preparation of some of the model input data.

   The VMod component can be left out if you don't need to use the VMod 2D hydrological model. This can be also installed later if needed.

   When you have selected the components you want to install, press **Next >>** to continue.



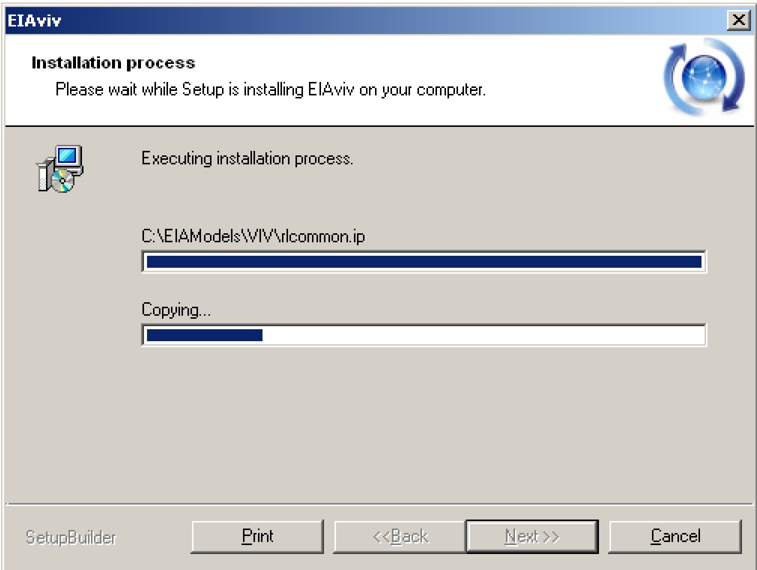
6. **Choose destination location:** Select the location you want to install the model software. The suggested directory [C:\EIAModels\VIV] is recommended to be selected as it is also used in this manual as the reference directory. Thus, by selecting this directory it would be easier to follow the manual as well. Also, some parts of the model and data processing don't support spaces in the file names and thus, it is not recommended to install the programme for example under the "programme files".

   If you wish to change the directory, press **Browse** and select the folder and location you want to install the software to.

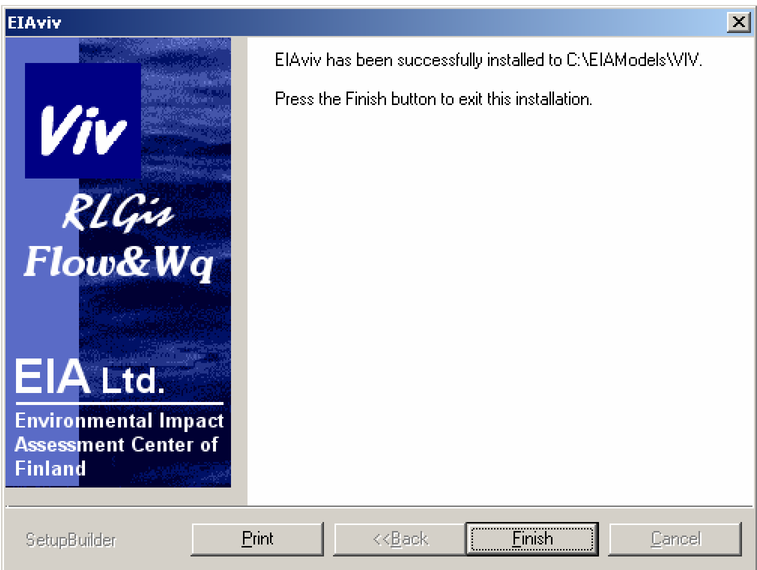
   Press **Next >>** to continue.

7. **Start installation:** You can check once more the target directory and user information to be sure that everything is ok before starting the installation. Press **Next >>** to start the installation.

   You can follow the installation process from the **Installation process** window.



8. If the following window appears, the EIAviv has been successfully installed to your computer. Press **Finish** to exit the installation.



If the model software setup is not working or there comes an error message the reason can be following:

- Some of the Viv-software components are running.
  → check this by going to Task manager and there processes. If you see a process of Viv, end that process and try to install the software again
- You haven't had the administrator rights when installed the software
  → sign in to the computer as administrator
- You don't have enough space in your hard disc
  → release space in your hard disc and try to install again

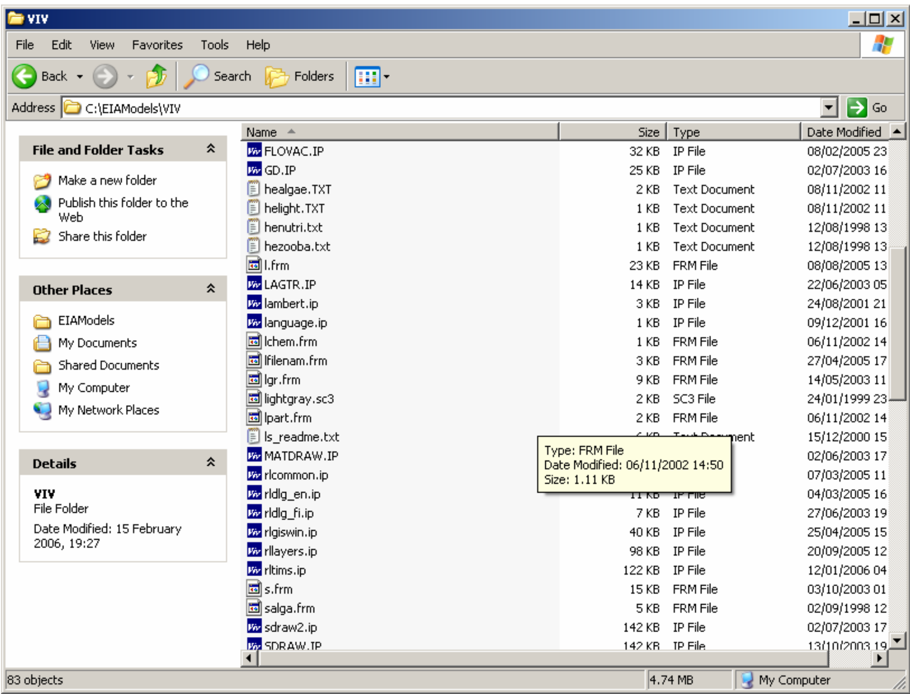
3.2 FILE SYSTEM

The EIAModel system consist two groups of files:

- model system files
- model application files

3.2.1 Model system files

The model system files are installed during the model software installation (Section 3.1 - Software installation) and are typically located under C:\EIAModels\VIV folder. The model system files contain the models and user interface programs.



Don't make any changes to the files in VIV-folder nor move these files or the software may not work properly anymore.

3.2.2 Model application files

The model application files define the actual model application grids and contain also model related data. These files are installed in C:\EIAModels directory under each model application sub-directory. Illustration below shows the model application directory structure.

C:\EIAModels

VIV – model system files

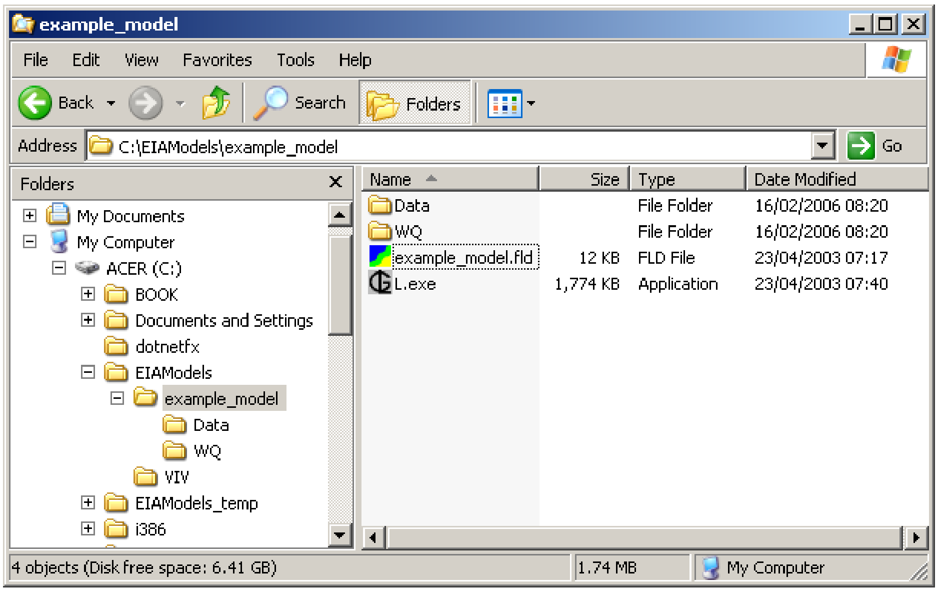
Example_model – model application files

Data – time series data files folder

WQ – computed results folder

example_model.fld – flow model parameters

L.exe – model executive file



Make sure that the L.exe is in the same folder than the model application file (*.fld – file). In case it is in different folder, its path has to be defined properly in Source data – Application setup part of the menu.

Models can be started by opening a preferred model directory under the application directory, and double clicking a model parameter file. For example, to start the example_model open the C:\EIAModels\example_model directory and double click "example_model.fld" file (this is a flow model parameter file).

File types listed below are found in the model application directories. Double clicking files marked with **underlined bold** type in windows explorer will start associated model user interface with the selected file.

Flow model directory

- ***.fld** – flow model parameters
- ***.wqd** – water quality model parameters

Data file types (for example in data-subdirectory)

- *.txd – timeseries data files (see txd format below)
- *.ipd – RLGis geographic data

Wq subdirectory

- *.gra – flow model timeseries data
- *.tek – flow model animation data
- *.out – flow model run text output
- *.exv – flow model output field
- *.exc – water quality model end field
- *.dat – user interface generated model input files
- filenam - user interface generated water quality model file list
- lfilenam- user interface generated flow model file list
- param.inp, nest*.dat - user interface generated nest data
- *.err – flow or water quality model error report

3.2.3 TXD file format

TXD file format a text file format for storing structured table data. The file contains contain two parts, the file header and the file data. The file header contains any number of file identification information and a data definition. The data part contains any number of data rows divided into data fields as defined the file header.

```
txd2
identification line
identification line
…
field definition
field definition
…
data
data line
data line
```

The file should always start with a row containing the text "txd2". After this line are the file identification lines, where each line consists of an item identifier and an item value. The item identifier must be a string starting with a letter and containing no spaces or tabs. The item value can be a number or a string. For example,

```
location "Koijärvi 2"
ypos 65.236
xpos 22.183
```

Additional identifiers, such as data source, missing value identifier and coordsys explanation can be added to further identify the data.

A field definition is composed of a field type, field name and field length/format. Following field types are available.

str - string
bool - integer 0-1
byte - integer 0-255
int - integer 32-bit
real - real number
time - time, e.g. date and clock values together
date - date value

Time type fields have a format definition instead of length definition. The format definition is a string in double quotes (“”) containing letters D,M,Y,h,m,s, meaning date, month, year, hour, minute and seconds, for example “YYYYMMDDhhmmss”.

```
time date “YYYYMMDD hhmm”
```

After the field definitions there is a line containing the word “data”, and after this the data lines. Data values must be always separated by at least one space character, the field lengths do not include this space character. Below is an example of a complete txd file. String fields in the middle of the row may not contain spaces within the string, for the last field of the row this restriction does not apply.

```
txd2
location "Lake A 2"
xpos 22.183
ypos 65.236
missing -9999
time date "YYYYMMDD hhmm"
int wdir_degr 3
real wspeed_m/s 5
DATA
19990101 1200 234 5.0
19990102 1200  34 3
19990103 1200  45 35.0
19990104 1200  43 45.0
```

Summary of standard file ids:

location –	data location name
statid –	station identifier (usually 4 character long string is used)
xpos –	measurement point x – coordinate, can be UTM or longitude
ypos –	measurement point y – coordinate, can be UTM or latitude
coordsys –	optional, defines the coordinate system for xpos and ypos variables.
dbname –	optional, defines measurement data file in measurement point file.
pidfile –	defines measurement point file in measurement data file
missing –	identifies missing data value used in file

Summary of standard field names:

date –	contains time value of measurement
depth –	contains measurement depth in meters
variable –	contains variable code if several variables are in same file
value –	contains variable values, used together with "variable" field
pid -	contains a unique identifier for the location, usually a 4 character long string is used (e.g. BAT1, KCH3)
xpos –	in measurement point file, contains point x-coordinate
ypos –	in measurement point file, contains point y-coordinate
name –	in measurement point file, contains location name

Summary of standard codenames and units for variables:

PREC_mm	precipitation
WDIR_degr	wind direction
WSPD_m/s	wind speed
FLOW_m3/s	discharge
WLEV_mm	water level
EPAN_mm	pan evaporation
TAVG_C	daily average air temperature
TMIN_C	daily minimum air temperature
TMAX_C	daily maximum air temperature
RHUM_%	relative humidity
TWMI_C	daily maximum water temperature
TWMX_C	daily maximum water temperature
PRES_kPa	pressure
DEPTH_m	measurement depth
DINN_ug/l	dissolved inorganic nitrogen
DIPP_ug/l	dissolved inorganic phosphorous
TOTP_ug/l	total phosphorous
TEMP_C	temperature

PHVA_lgH	pH
SEDI_mg/l	suspended sediment
COND_mS/m	conductivity
CA_mg/l	Ca
MG_mg/l	Mg
NA_mg/l	Na
KA_mg/l	Ka
ALK_mmol/l	alkalinity
CL_mg/l	Cl
SO4_mg/l	sulphate
NO2_ug/l	nitrite
NO3_ug/l	nitrate
TOTN_ug/l	total nitrogen
NH4N_ug/l	ammonium
PO4P_ug/l	phosphate
SI_mg/l	Si
OXYG_mg/l	dissolved oxygen
OREL_pros	relative oxygen concentration (in percentage) compared to saturation value (0 – 100 %)
CODM_mg/l	Chemical Oxygen Demand (using Mn)
CHLA_ug/l	chlorophyll A
CDIR_cm/s	current direction
CSPD_deg	current speed
TURB_NTU	turbidity [Nephelometric Turbidity Units]

3.3 Hardware and operating system requirements

Hardware requirements for the model runs depend strongly on the application. The smallest test and training cases can be run on practically any PC regardless of the power of the processor or amount of memory. On the other hand especially applications used for bank erosion with large number of grid cells, small grid and time step size and turbulence and momentum advection calculation can run very slowly on less powerful hardware. Recommended hardware for large problems is:

- central memory 2 GB (even if model calculation loop will run in most practical cases without need for slow virtual (hard disk) memory even under 1 GB, there should be sufficient memory for other applications)
- dual or quad core processor (mostly to run efficiently long model runs while working with other software)
- AMD 64 X2 Athlon 5200+ or higher or Intel Intel Core 2 Duo E8200 (6 MB L2 cache, FSB 13333 MHZ) or higher.

Traditionally AMD processors have been faster for fluid mechanics simulations because their floating point and memory systems have been faster. Later Intel has improved memory speed and competes on par with or even exceeds AMD.

Multi-core processors have the potential to run parallelised programme code much faster. Although parallelisation has been implemented in the model, the latest model developments have not yet been checked with parallel execution. This is an option in the future to run large and complex applications faster.

Speed of video hardware is not an issue for the model runs. In some older video cards hardware acceleration causes model OpenGL interactive graphics to freeze. In this case hardware acceleration should be be reduced to basic setting.

Extremely large applications such as Lower Mekong Basin model with less than 1 km grid size may not be able to run under 32 bit operating systems because they exceed 2 GB per-process memory limit. Model should then be compiled with a 64 bit compiler.

64 bit operating systems offer also potential for faster code execution because of among others large number of utilised registers. However, the limited test runs with 64 bit Windows and Linux have been slower than on 32 bit Windows. This may be because the compilers used in the tests were not optimal.

4 BASICS FOR USING EIA 3D MODEL SYSTEM

In this chapter the basics for use of EIA 3D model system are presented including starting the model, graphical user interface, structure of the main menu and tool-bar. The opening and saving of existing model application are also described.

The chapter is divided to five parts:

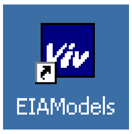
4.1 Starting the EIA 3D model software

There are two options to start the EIA 3D model application:

- From the EIAModels shortcut icon on the desktop
- By open the model application and at the same time the model software by open the *.fld file under the C:\EIAModels\[model_application_name] directory

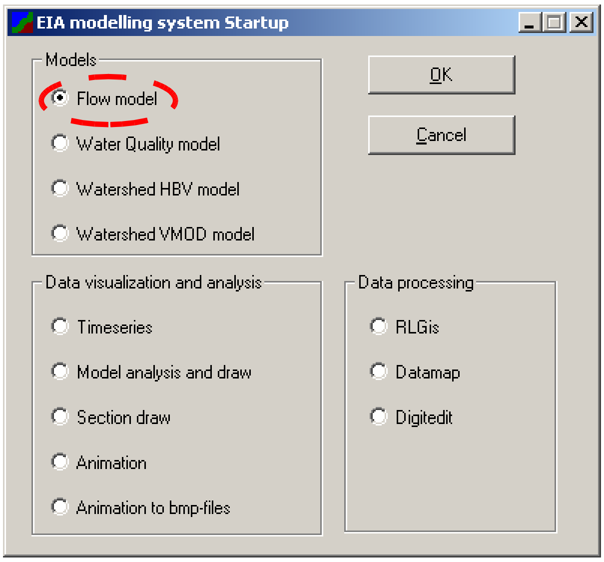
4.1.1 Starting the software from EIAModels desktop shortcut icon

1. double click the EIAModels shortcut icon on your desktop

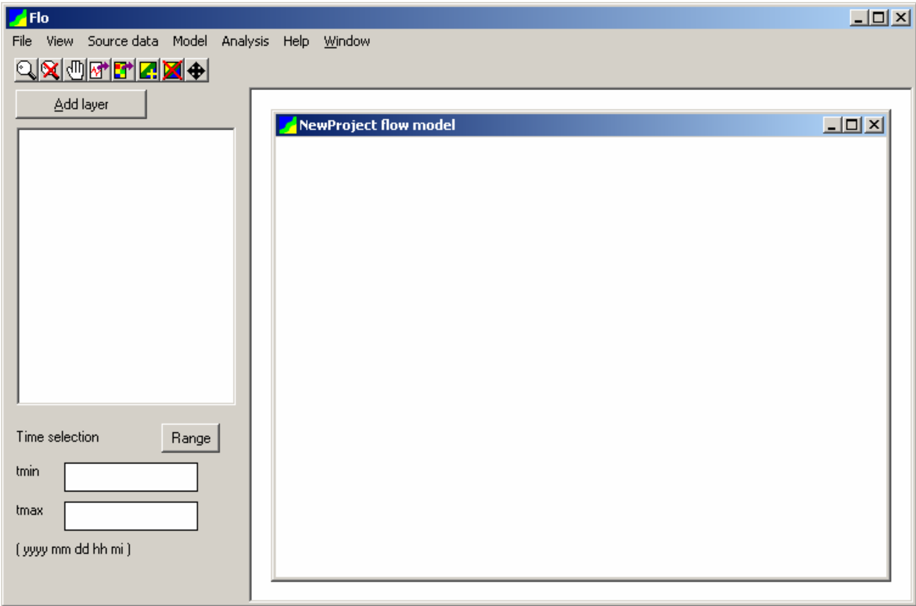


If the EIAModels shortcut icon doesn't exist in your desktop there are two options:

- Your installation hasn't been completed successfully
  → re-install EIA model software to your computer
- Someone has removed the shortcut icon from your desktop
  → go to folder C:\EIAModels\VIV
  → click right mouse button the file named EIAModels.ip
  → select **Create shortcut** and the shortcut is created to the folder you are in
  → copy/cut/drag the shortcut into your desktop

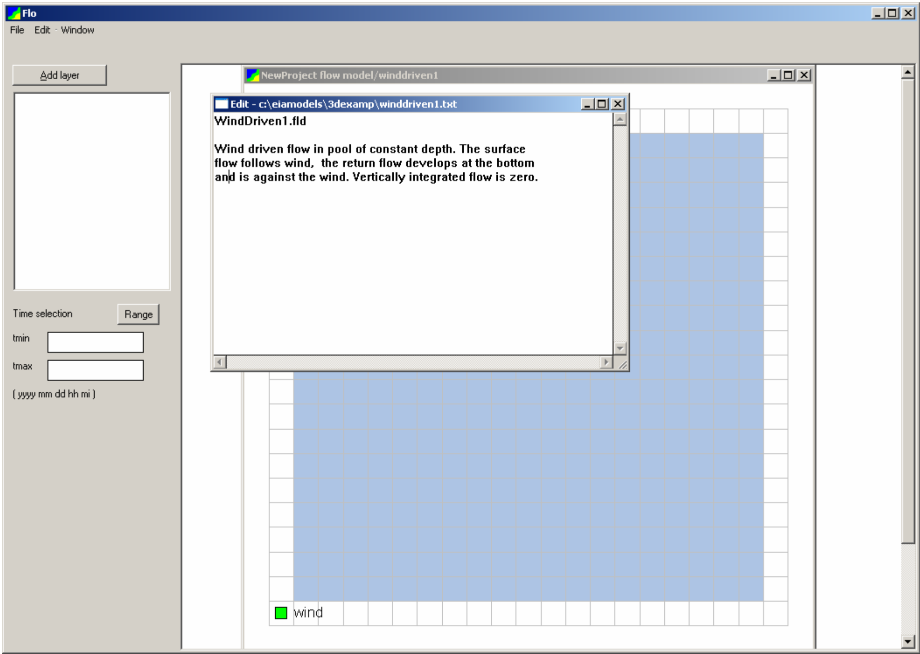


2. Select the flow model and press **OK**
3. The Flo-window opens with the main menu, tool-bar and NewProject flow model window



4.1.2 Starting the software from fld-file

1. go to the directory of the model application you want to open (e.g. C:\EIAModels\3DEXAMP)
2. double click the model application you want to open (e.g. winddriven1.fld)
3. The model software opens with the model application you wanted to open



If the EIAModels doesn't open as shown above, there are few options to check:

- Your installation of the model software hasn't been completed successfully
  → re-install EIA model software to your computer
- During your installation you haven't been logged in as an administrator and part of the software registration hasn't been finished and thus, the software doesn't work properly.
  → log-on to the computer as Administrator and install the model software again

4.2 Graphical User Interface (GUI)

The flow model user interface main window is shown below. The window contains a menu, a toolbar and a work area. The menu is used to select actions to be performed. The toolbar contains, for example, tools for moving around in the model grid area. The work area may contain different type of windows, for example model grid window, time series windows and data table windows (Figure 9).

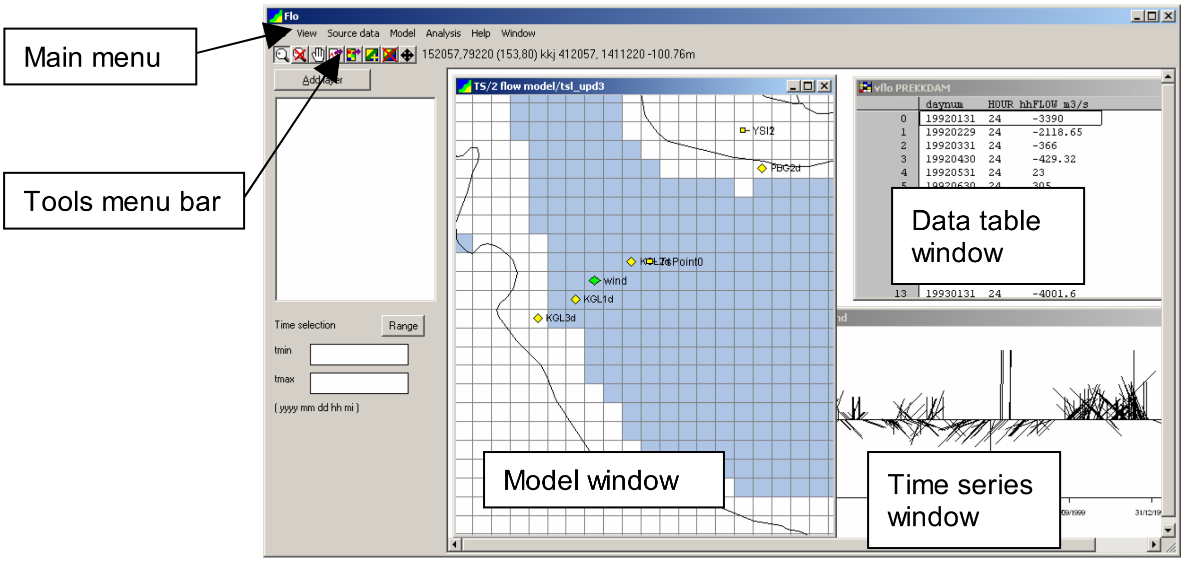
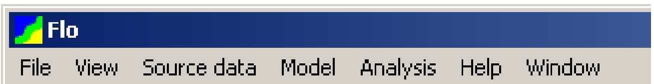


Figure 9. EIA 3D Model graphical user interface.

4.2.1 Main menu

The menu structure of the model reflects model usage, which can be roughly divided into seven steps:

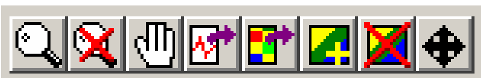
1. File handling and model grid importing, File menu
2. View control options, View menu
3. Input data definition, Source data menu
4. Setting model parameters and Model computation, Model menu
5. Examination of results, Analysis menu
6. Help functions, Help menu
7. Window control, Window menu



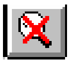
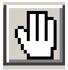
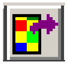
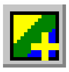
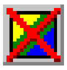
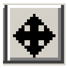
The main menu bar and tools menu bar change according to the active window.

4.2.2 Tools menu bar

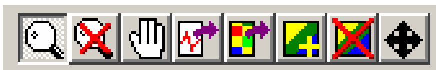
The tools menu bar shows several tools related to picture handling and data item management.



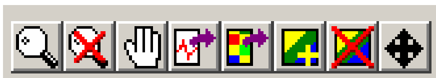
The tools are described in more detailed below:

Tool	Function
	**Zoom:** Magnify or zoom into the picture. Click the button and then drag a rectangle with mouse to the window.
	**Zoom out:** Maximum view or Zoom out. Click this button to return to the no zoom state. No effect if no already in maximum view.
	**Move:** Pan the map
	**Copy as metafile:** Copy the picture to clipboard as a picture (Windows metafile). Can be used to transfer pictures to text processing and drawing programs.
	**Copy as bitmap:** Copy the picture to clipboard as a bitmap (Windows metafile). Can be used to transfer pictures to text processing and drawing programs.
	**Add item:** Add an item (flow, timeseries, concentration, load, etc) to the map
	**Remove item:** Remove an item from the map
	**Move marker:** Move item in the map

To release the selected tool press right mouse button. You can see the selected tool as pressed down. E.g. zooming tool is selected in illustration below:

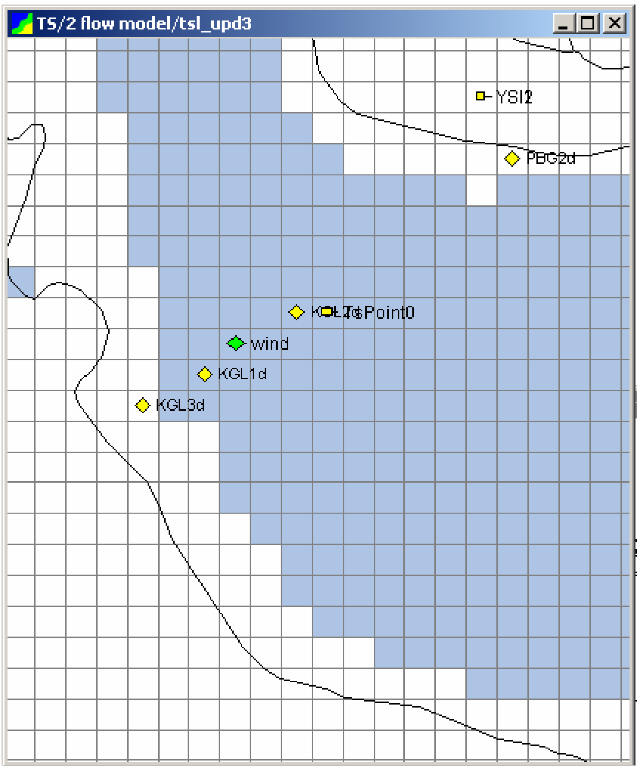


→ click right mouse button and the zoom is released as shown below



4.2.3 Model window

The model window shows the computational model grid, where usually water is displayed with white and land with white colour.



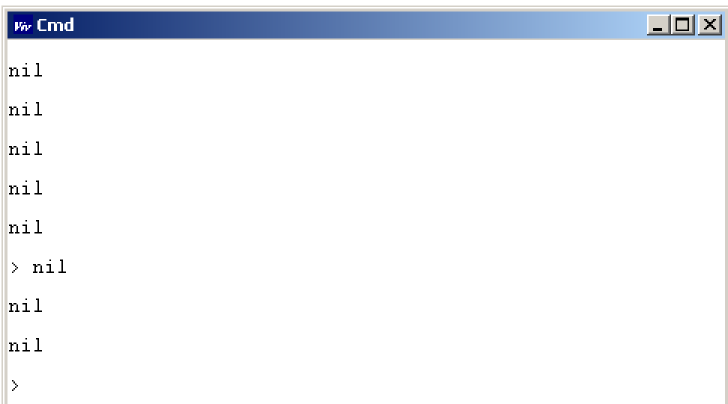
The grid is decorated with symbols representing different model input and output data items, for example:

- Time series site (output)
- Discharge site (input)
- Flow boundary value (input)
- Wind (input)

4.2.4 Command window

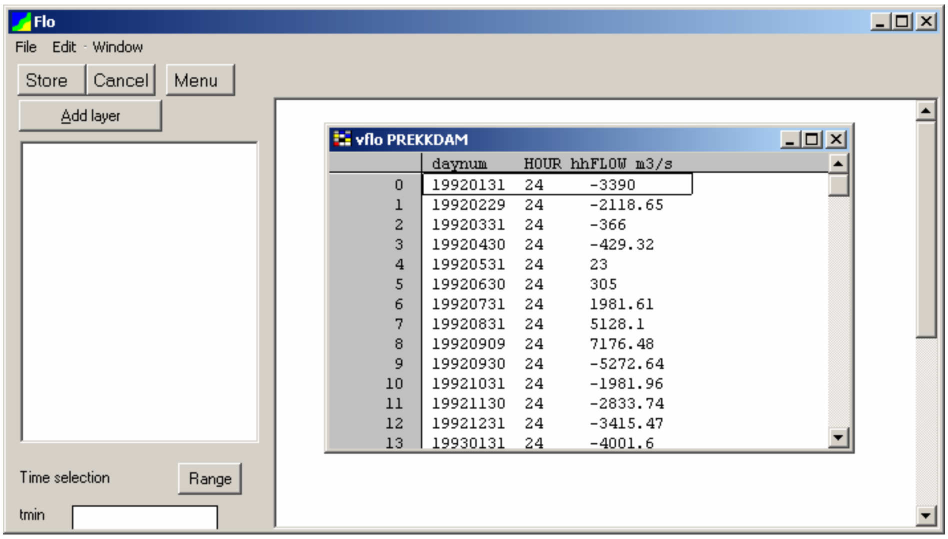
Command window informs the possible errors within the model use. User can also give commands to the programme if needed. Most of the commands, however, will be given through the GUI and Command window provides mostly information from the programme to the user.

You can open the Command window from **View – Show Cmd window**



4.2.5 Data table window

The model input data, as discharge measurements, water level, etc, can be accessed and/or edited and analysed through the data table window:



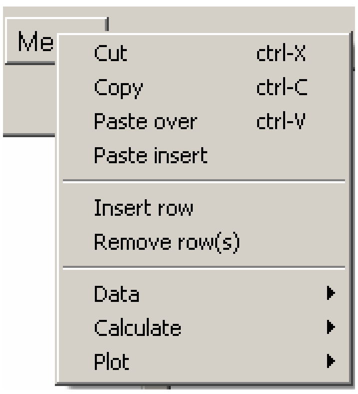
The table itself shows the date of the data and data itself. As can be seen the Menu bar and toolbar have changed from the one which is available when model window is open.

The user is able to change the values in the table if needed.

Don't change the boundary condition data if you are not sure that the change is needed e.g. then when some data is wrong. Changing the boundary condition data values may impact on the model results.

If you do some changes to data table which you want to keep, you must always save the changes by pressing **Store** on the toolbar.

When clicking the **Menu** the following menu appears:



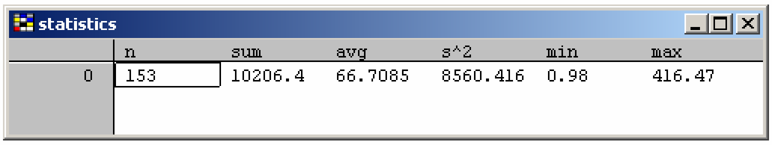
From the Menu button on the toolbar you can do following actions:

- Cut ctrl-X
- Copy ctrl-C
- Paste over ctrl-V

- Paste insert
- Insert row
- Remove row(s)
- Data
    - Create matrix
    - Write to file
    - Insert to file
- Calculate
    - Statistics
- Plot
    - XY dot
    - XY line
    - Area
    - Line
    - Bar

By using the tools under **Data**, one is able to create new matrix, write that to the new file or insert to the existing file.

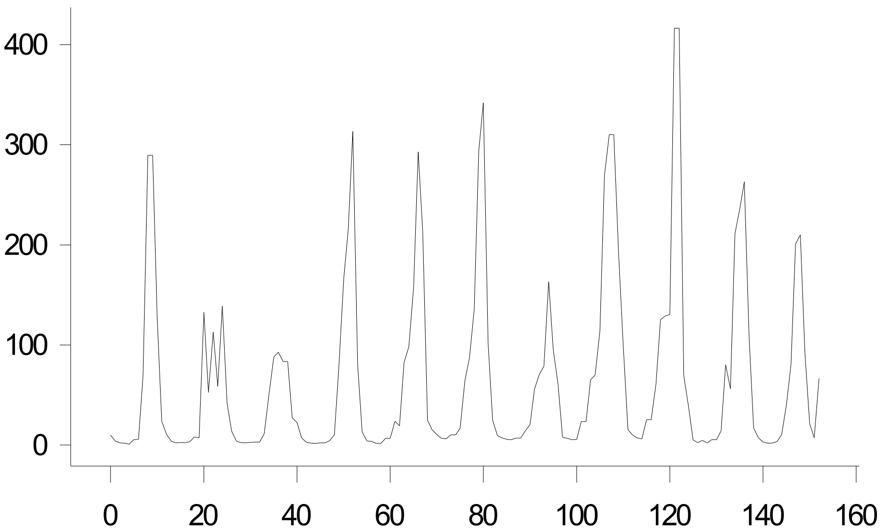
From **Calculate – Statistics** the basic statistics such as number of samples, sum, average, standard deviation, and minimum and maximum values. The result from one dataset is shown below:


statistics

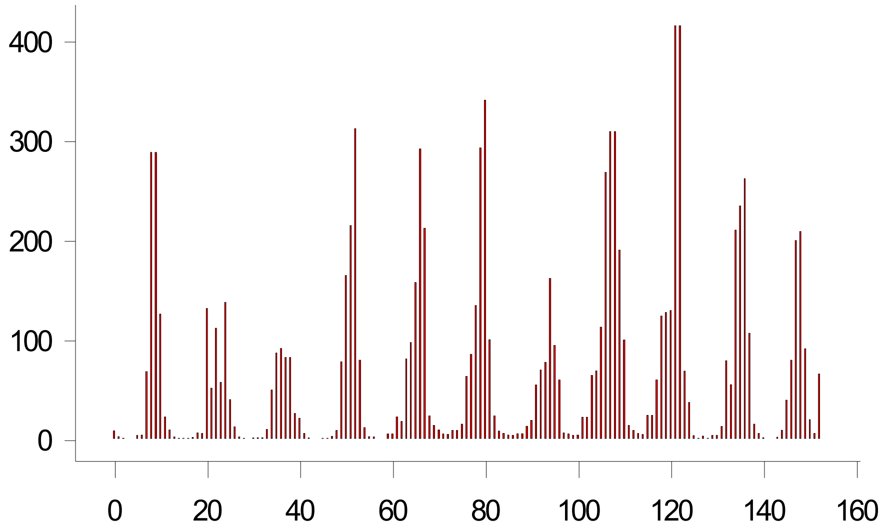
	n	sum	avg	s^2	min	max
0	153	10206.4	66.7085	8560.416	0.98	416.47

With the **Plot** tools the user is able to illustrate the data in graph as illustrated with few examples below:

Plot – Line:



Plot – Bar:



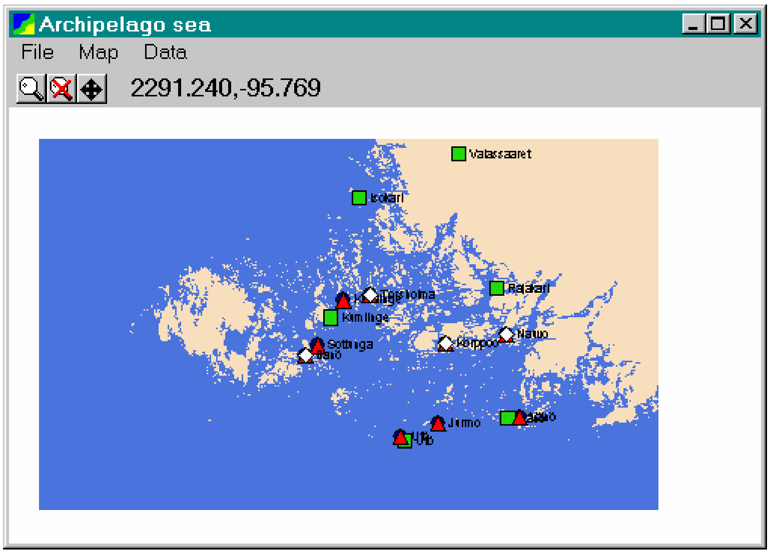
4.2.6 Timeseries window

Managing of timeseries window is explained in Section 15.1 – Picture timeseries.

4.2.7 DataMap window

DataMap is used to handle flow model related measurement data DataMap window consist of a digitised base map and a set of symbols representing measurement points. DataMap-feature has not been utilised in the Mekong applications and it is not present in the latest versions of the user interface. It has been replaced by a more versatile RLGis-software.

You can open the DataMap from **View – Show DataMap**



Measurement data can be displayed by clicking a symbol on the map using mouse.

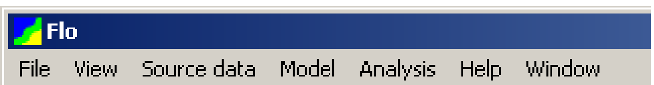
Measurement points are shown over the map base as symbols. Each symbol is connected to one measurement time series that contains values of a measured variable in a location. Displayed symbols can be selected using a window that is accessed using **Data/Visible data** menu

The visible map information (map layers) can be selected to display, for example, shoreline and water and land areas in different colours. The selection of map layers is done through **Map/Map layers** menu.

4.3 Menu structure

The menu structure of the model reflects model usage, which can be roughly divided into seven main component and each of those to sub-component as listed below. The page numbers from where more information for each menu item can be found are listed in Table 1.

1. File handling and model grid importing, **File** menu
    a. Application file handling
    b. Other
2. View control options, **View** menu
    a. Zoom control
    b. Window options and management
3. Input data definition, **Source data** menu
    a. Boundary condition data management
    b. Timeseries data management
    c. Application setup
4. Setting model parameters and Model computation, **Model** menu
    a. Grid parameters
    b. Model parameters
    c. Model result parameters
    d. Model computation
5. Examination of results, **Analysis** menu
    a. Field results
    b. Animation results
    c. Timeseries results
6. Help functions, **Help** menu
7. Window control, **Window** menu
    a. Window management
    b. Application windows



The structure of the main menu is illustrated in details in Figure 10. Each component, sub-component, and menu item of the Main menu with the corresponding page(s) in the manual where the functions of menu item is explained is presented in Table 1.

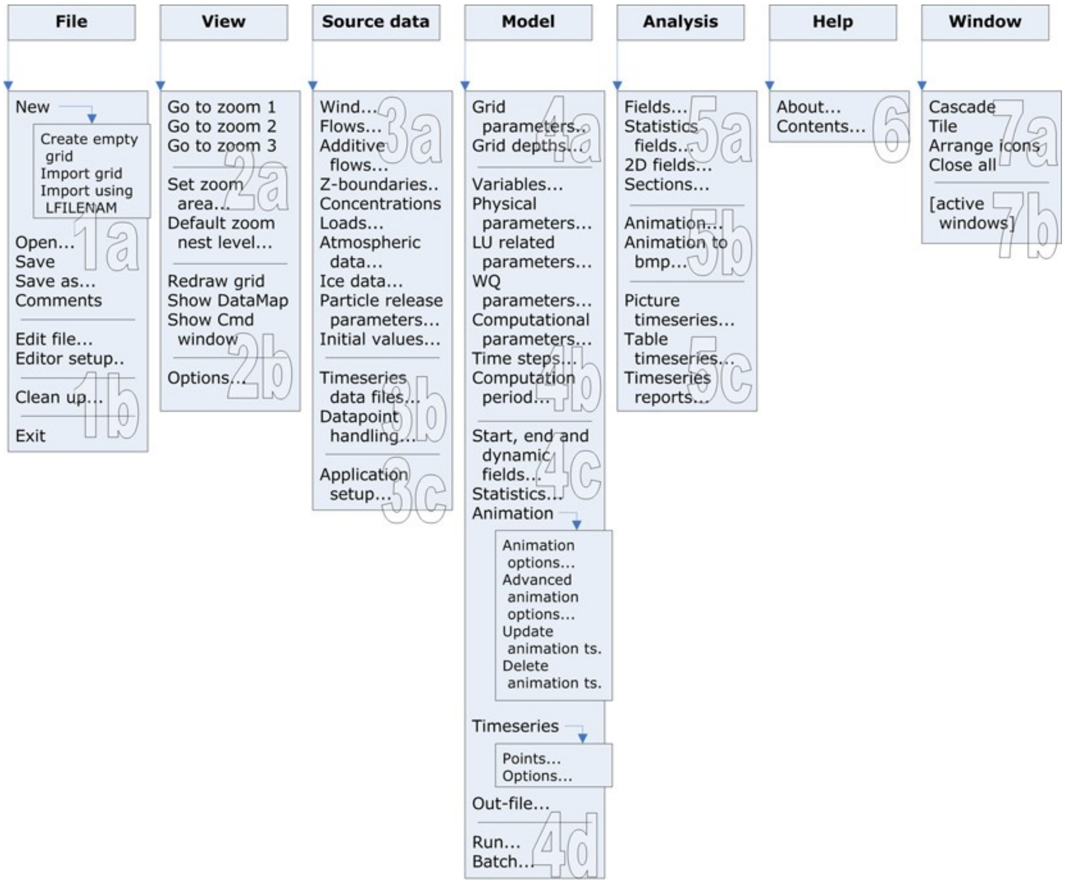


Figure 10. Structure of the main menu (the names of the sub-components are listed in previous page).

Table 1. Each component, sub-component, and menu item of the Main menu with the corresponding page(s) in the manual where the functions of menu item is explained.

Main component	Sub-component		Menu item	Page
1. File	1a	Application file handling	New -->	64
			Open…	60
			Save…	61
			Save as…	61
			Comments	
	1b	Other	Edit file…	
			Editor setup…	
			Clean up…	
			Exit	61

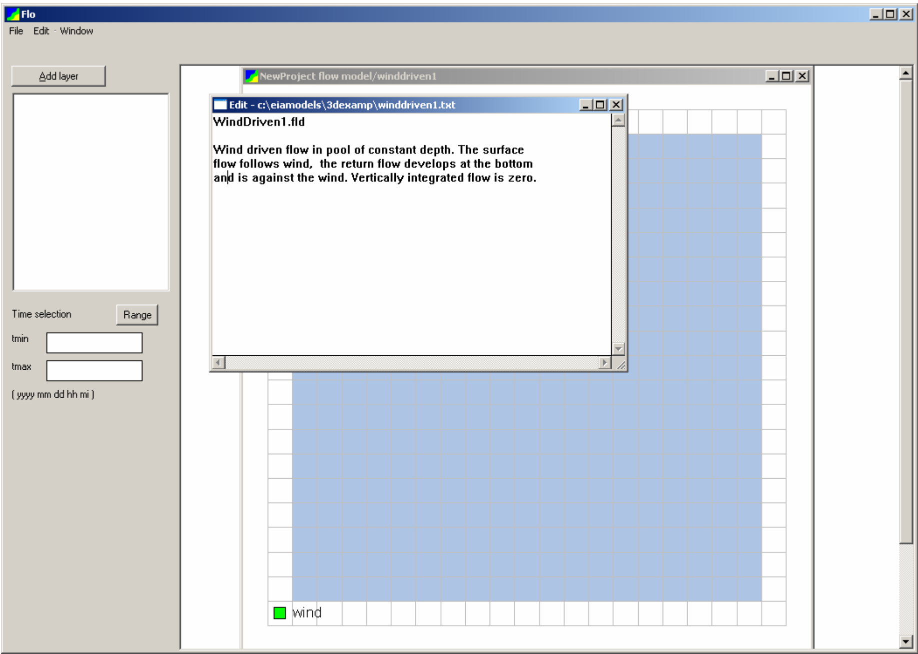
6. Help			About…	222
			Contents…	222
7. Window	7a	Window management	Cascade	224
			Tile	224
			Arrange icons	224
			Close all	224
	7b	Application windows	[active windows]	224

4.4 Open existing model application

4.4.1 From fld-file

One option to open existing model application is to open it directly from the model application file (*.fld). as explained below:

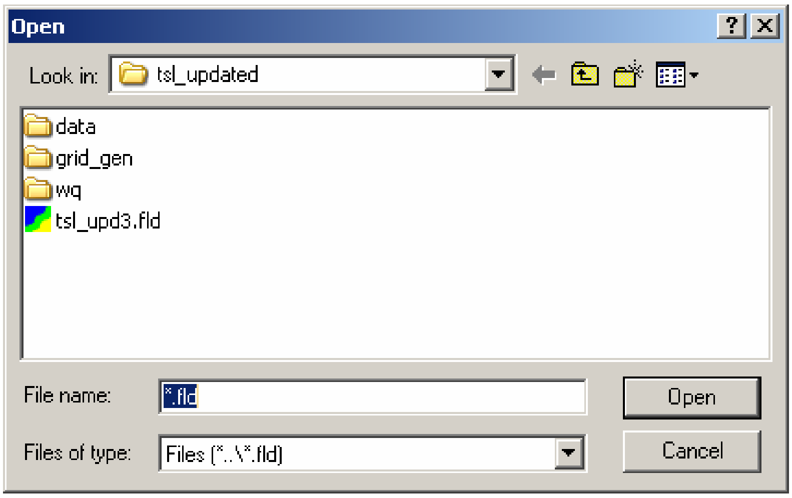
1. go to the directory of the model application you want to open (e.g. C:\EIAModels\3DEXAMP)
2. double click the model application you want to open (e.g. winddriven1.fld)
3. The model software opens with the model application you wanted to open



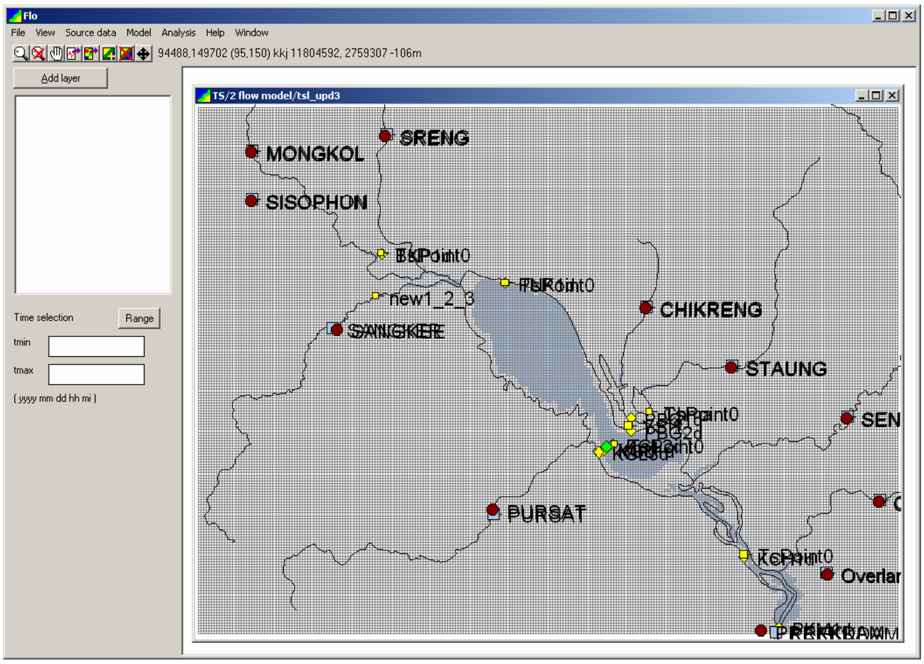
4.4.2 From GUI

Second option to open the existing model application is to open it from the model GUI if the EIA 3D model is already open.

1. From main menu select **File – Open**

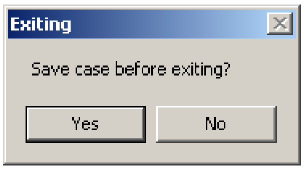


2. go to the folder where the model application is and select the model application file (*.fld) you want to open
3. press **Open**
4. The model software opens with the model application you wanted to open

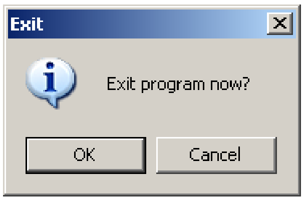


4.5 Exit model application

To exit model application select **File – Exit** from main menu. The following window appears:



If you want to save the model application before exiting, press **Yes**, otherwise press **No**. After that model still asks whether you are sure to exit the model or not. Press **Yes** to exit the model. If you want to resume, press **No**.

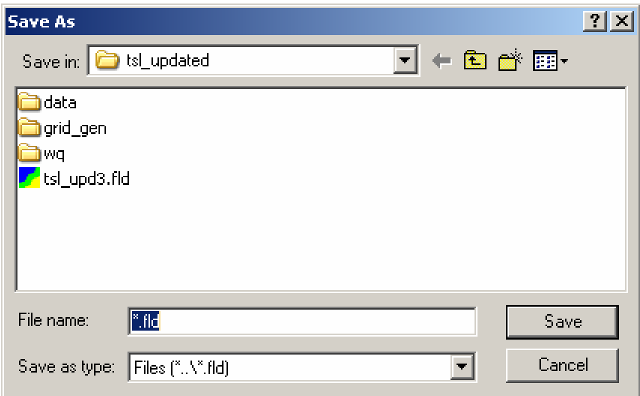


4.6 Saving model application

4.6.1 From GUI

To save your model application and all the settings in it, follow the instructions below:

1. From main menu select **File – Save** or if you want to save the application with different name **File – Save as…**
2. If you selected **Save** and you have already saved the application before, the model is saved like that. If you selected **Save as…** or **Save** and the model is not saved before (new application) the following window appears:

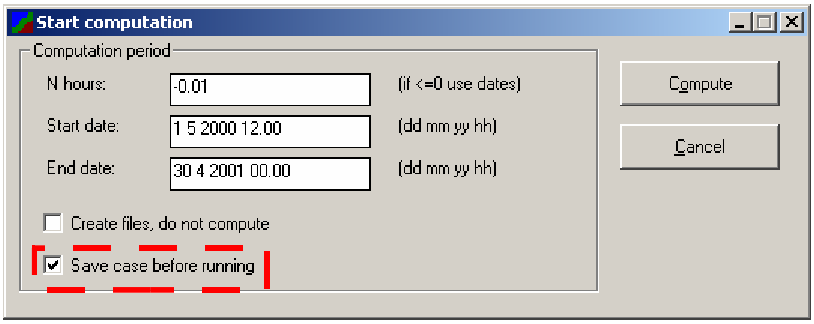


3. Select the destination folder and the name you want to save the model application with
4. Press **Save** and the model is saved

4.6.2 Save case before running

There is option also to save the model application each time before running it.

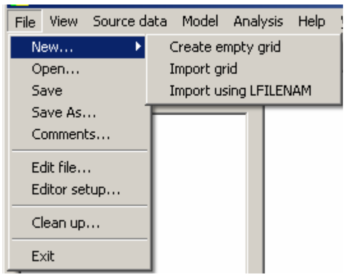
1. From Main menu go to **Model – Run…**
2. Select on the **Save case before running** - box



3. Press Compute to run the model

You are not able to undo the saved model settings. Thus, if you are making big changes on the parameters or other model settings, it is always good to keep the original model application saved with another name (see above **Save as…**).

5 CREATING NEW MODEL APPLICATION



This chapter introduces how to create a new model application either from elevation data or manually by creating empty grid.

The chapter is divided to two main parts:

5.1 Creating empty grid

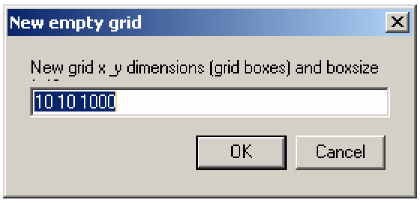
5.2 Creating model grid based on elevation data

5.1 Creating empty grid

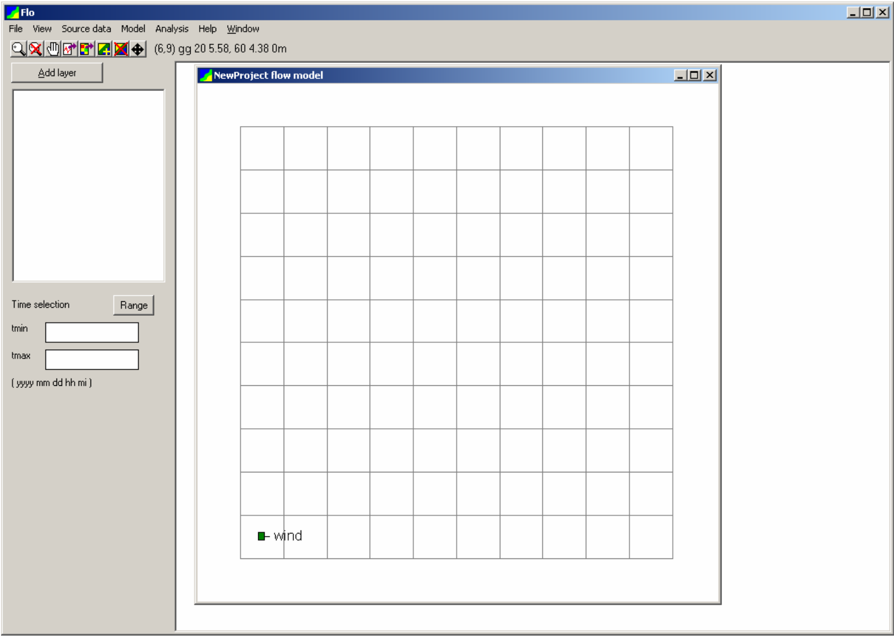
The model application can be created either based on the elevation data (next section) or creating empty grid which is dealt in this section.

1. Select **File – New – Create empty grid**

2. Press **OK**
3. Enter the dimensions of the grid and box size in meters following the guidelines: x-dimension y-dimension box_size (as a default the grid is 10*10 boxes with the box size of 1000 m). The number of your application depends, of course, the application you want to create.



4. Press **OK**



5. The empty grid is ready.

> The maximum number of 3D boxes (x*y*z) that normal 32 bit computer (normal PC) can handle is around 5'000'000. Thus, if you are using for example 10 vertical boxes (for more information, see Section 8.1.5 – Vertical division), the maximum number of horizontal boxes is around 500'000 (e.g. 500*1'000).

There are three options to define the depths of the grid:

- Manually using the **Model - Grid depths, Map** (for more information, see Section 8.2.1 – Modifying the grid depths in map) tool to define the depth for each grid cell
- Manually using the **Model – Grid depths, Table** (for more information, see Section 8.2.2 – Modifying the grid depths in table) tool to define the depth for each grid cell
- From table where depths are already in place by copying the information to the model by using the **Model – Grid depths, Table** (for more information, Section 8.2.2 – Modifying the grid depths in table) tool

> **To select the optimum grid size**
> The model grid size (box size) impact on the model accuracy, natural

5.2 Creating model grid based on elevation data

To create the grid the original data can be in many different forms: digital elevation model (DEM), depth points, and/or contour lines. The files can be also in many different file formats: *.shp, *.bil, *.ipd, *.dig, etc. Thus, you should have some basic knowledge of the GIS in general and the basic use of some common GIS applications should be known as well.

5.2.1 Data needed

The data needed to create a grid for 3D Model are basically any data from the elevation of the ground surface: DEM in *.bil or some other format, contour lines in *.shp or *.ipd format, depthpoints in *.shp, *.ipd, or *.dig format. From the data you have the first phase is to create a *.dig file of the original data you have.

5.2.2 Data conversion

In this part some of the most common data conversion will be presented. It is impossible here to introduce all the possible alternatives. Hence, to do more complicated data conversions there could be a need to assistant from the trainer.

Original file in *.bil format

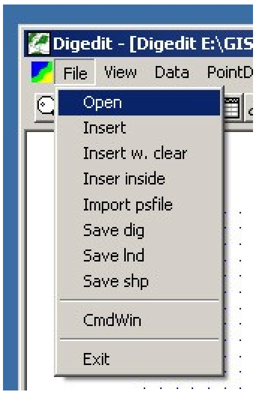
When the original DEM file is in *.bil format: Open the bil-file in RLGis and save it to *.dig format.

1. Open RLGis program
2. Add layer – import file - *.bil
    a. browse the needed *.bil file
    b. open it
3. Do the needed operations (if any) as resize, change the format, change values, etc.
4. Save the file to *.dig format
    a. Click right mouse button on the file in the left file list.
    b. select save
    c. type the file name
    d. select the right format: dig
5. Go to the next step: Convert *.dig file to 3D grid

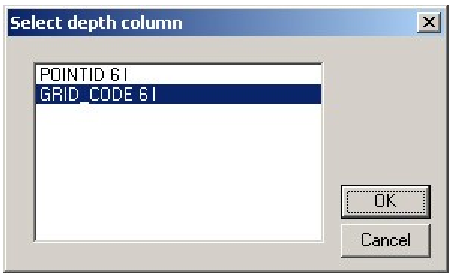
Original file in *.shp (point) format

When the original DEM is as depth point format in *.shp file: Open the *.shp file in Digedit and save it to *.dig format

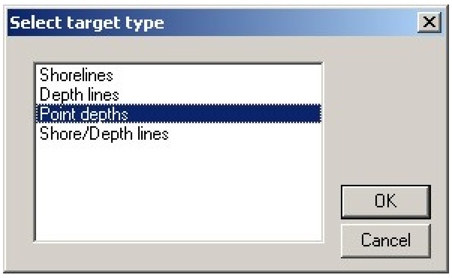
1. Open the Digedit program
2. Open the .*shp file



    a. for depth column you have to select the column in where the information of the elevation is. Normally that is called GRID_CODE (look image below).



b. for target type you have to select the point depths (look image below)



3. Save the file to *.dig format
    a. Select File – Save dig
    b. Type the name of the file
    c. Press Ok
4. Go to the next step: Convert *.dig file to 3D grid

5.2.3 Convert *.dig file to 3D grid

First you have to copy the convert program to the same folder where your final *.dig file is located. Hence, copy the following files

- higeneva.exe
- nestneva.exe
- coni
- cohi

to the same folder where the dig-file is. Those program files are e.g. in folder **gridgen.**

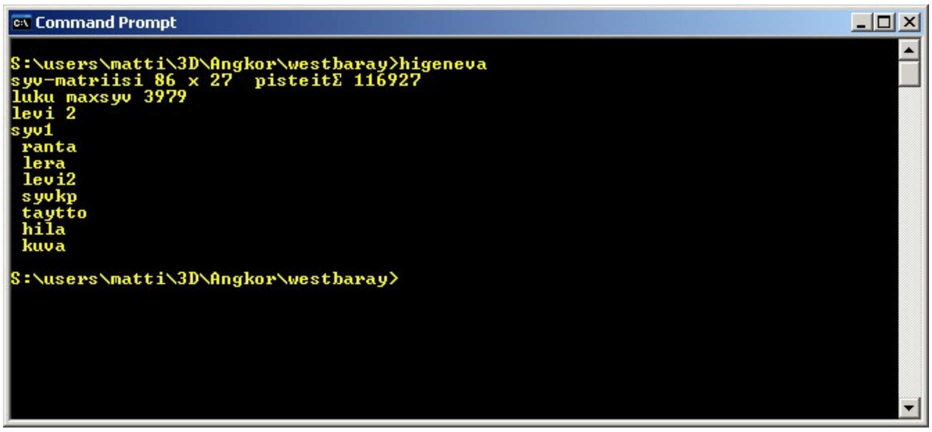
Higeneva

>> input: coni and digit.dat **<<**

1. rename the *.**dig** file created from the DEM to **digit.dat**
2. edit **coni** –file following the instructions below (using e.g. notebad):

0	\units: 0=meter, 1=latlong
0.001 1.	\horizontal conversion coefficient, vertical conversion coeff.
50	size of the grid box in chosen unit
1 356800 400000 1462800 1512000	\0=off, 1=on, minX, maxX, minY, maxY
0 360000 1341000	\0=off, 1=on, used point for determine the direction
0 2000 2500	\0=off, 1=on, parameters to the program

3. after the parameters are right in coni–file, run the program by typing to the command line: ***higeneva***



If the program doesn't run properly until the end (compare to the figure above) there could have been some problem in coni-file or then in final *.dig file. Check e.g. that the coordinates in coni-file are inside the area defined in *.dig file.

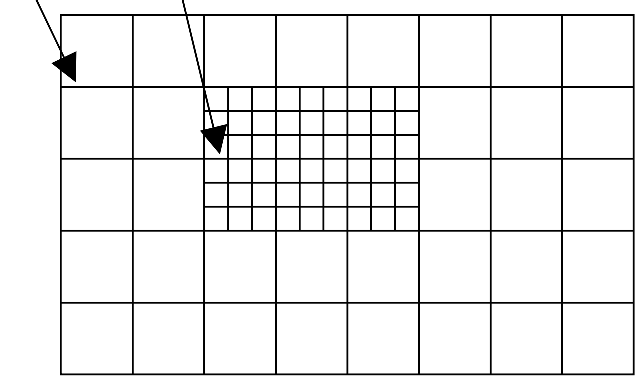
<< output: e.g. op **>>**

Nestneva

>> input: cohi and in1.dat **<<**

4. rename the **op** file to **in1.dat**
5. edit **cohi** –file following the instructions presented

1	/number of input data files (in1.dat, in2.dat etc.)
2	/number of subareas
50 357000 1463000 399800 1511000 10 368000 1482000 389800 1504000	/size of the grid box in meters, minX, minY, maxX, maxY (for both subareas!)



6. after the parameters are right in cohi –file, run the program by typing to the command line: ***nestneva***

<< output: op.ppm – graphical

grid.dat – for the flow model **>>**

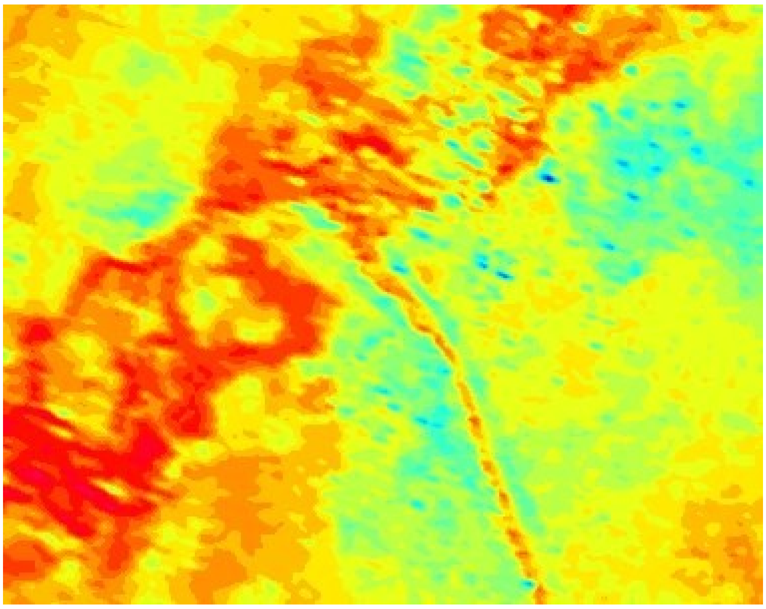
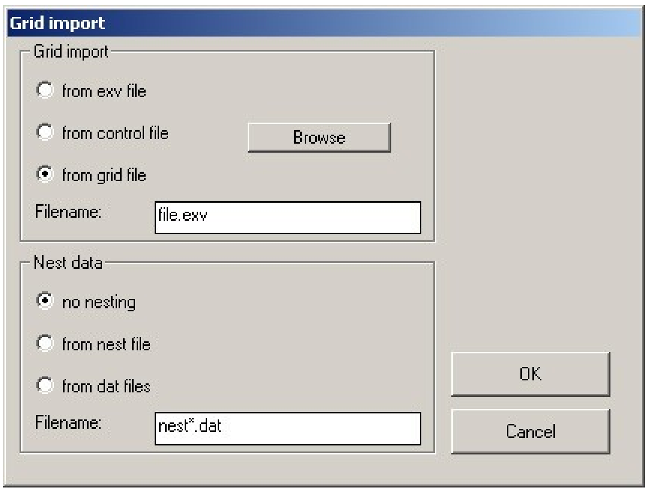


Figure 11. Example of op.ppm file – the colours show the altitude variation. Here red colour (dark) is low ground and green (light) colour is high ground.

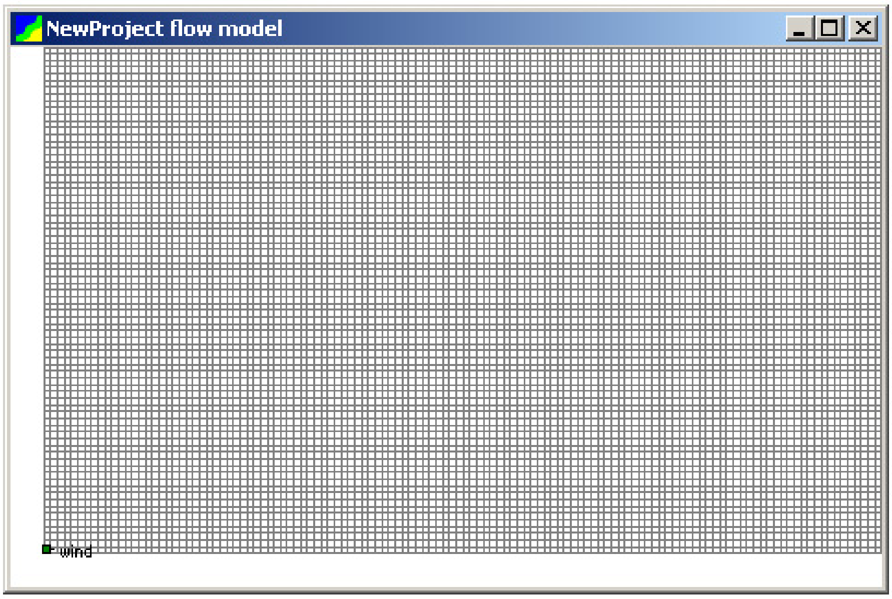
5.2.4 Import the grid to 3D model

To import the grid to 3D model you have to first open empty model and then create a new model by following the instructions below:

1. Open the 3D Flow Model (see Section 4.1 - Starting the EIA 3D model software on page 48)
2. Select from main menu **File – New – Import Grid** and click **OK**
3. In Grid import section select “from grid file”



4. Click **Browse**
5. Find the **grid.dat** file which you created in grid generation from .dig file and click open. The following window will appear

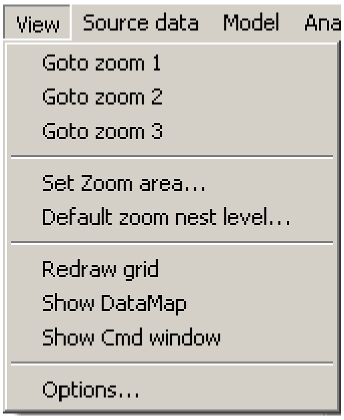


6. Save the model as explained in Section 4.5 - Exit model application.
7. Close the model application
8. Start the model application as explained in Section 4.1 - Starting the EIA 3D model software.

The grid is ready to be applied for the modelling.

> It is necessary to close the saved model application, once the grid is imported, and open it again that the model knows the right path to find the modelling related information as background maps, boundary conditions, etc.

6 VIEW OPTIONS



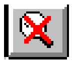
This chapter deals with the view options of the model graphical user interface.

The chapter is divided to five parts:

6.1 Zoom

6.1.1 Zoom and Zoom out tools

Zoom in: Magnify or zoom into the model grid or other picture. Click the zoom button  in the toolbar and then drag a rectangle with mouse to the window.

Zoom out: Maximum view or Zoom out. Click Zoom out button  in the toolbar to return to the no zoom state. No effect if already in maximum view.

6.1.2 Goto zoom

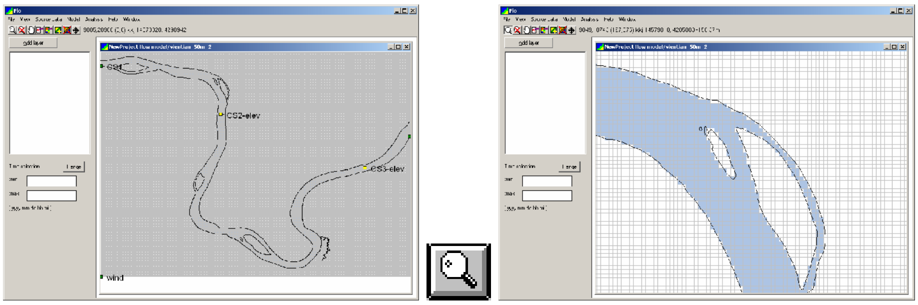
With "Goto zoom" command it is possible to go to the pre-defined zoom areas. Altogether three pre-defined zoom areas (1-3) can be set up. The instructions for how the zoom area can be set up are explained in the next section (6.1.3 - Set Zoom area…).

To go to pre-defined zoom area, select from main menu **View – Goto zoom [number of the zoom area you want to go (1-3)]**. The zoom area can be used also for defining the animation area (more in Section 10.3 – Animation options).

6.1.3 Set Zoom area…

With this command the user can set the pre-defined zoom area by following the steps below:

1. Zoom in to the area you want to set up for the zoom area by using the Zoom tool  in the toolbar.



2. Select **View – Set Zoom area…** from the main menu
3. Enter the Zoom number (1-3) to set this zoom area

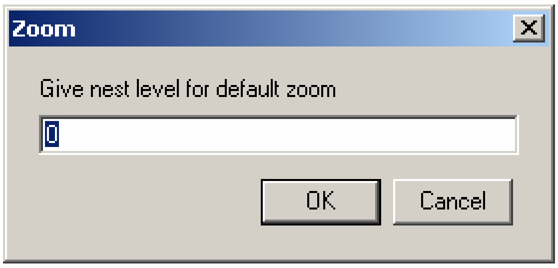
4. Click **OK** and the zoom area has been set

Afterwards to go to that zoom area, follow the instructions given in Section 6.1.2 - Goto zoom.

6.1.4 Default zoom nest level…

If you have nest levels in your model, you can zoom to those with this tool (if you don't have nest levels defined, you are not able to use this tool).

1. Select **View – Default zoom nest level…** from the main menu
2. Enter the nest level you want to zoom in (0 is the whole model area)



3. Click **OK**

6.2 Redraw grid

If you need to redraw grid, for example to see some updates you have made but which are not shown in the model window you can refresh the view by selecting **View – Redraw grid** from the main menu.

6.3 Show datamap

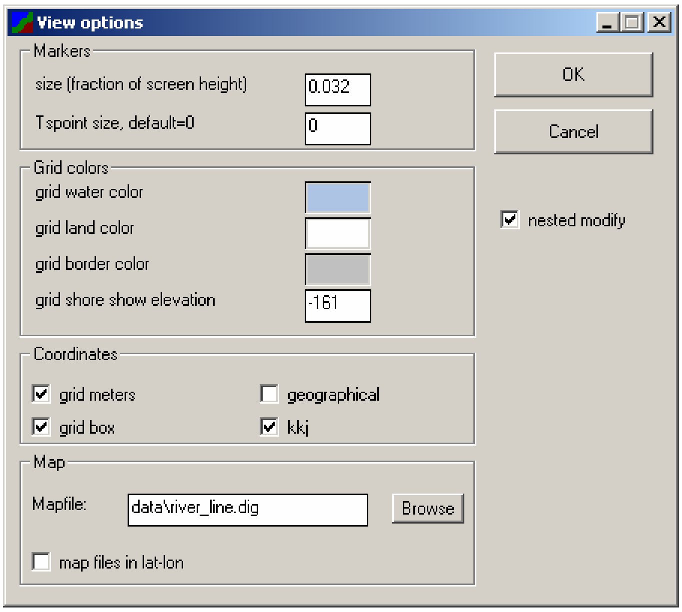
Select **View – Show datamap** from the main menu to show the datamap in the GUI. More about datamap can be read from Section 4.2.7 - DataMap window .

6.4 Show Cmd window

Select **View – Show Cmd window** from the main menu to show the Command window in the GUI. More about Command window can be read from Section 4.2.4 - Command window.

6.5 Options

With the view options you are able to manage the layout of the model window. Select **View – Options** from the main menu to see the options window.

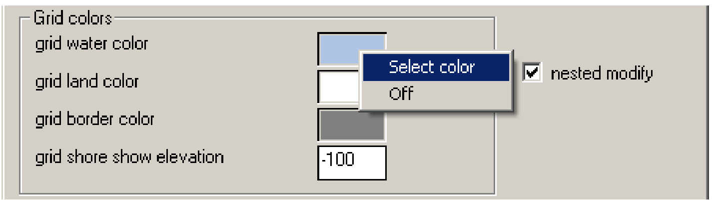


Markers

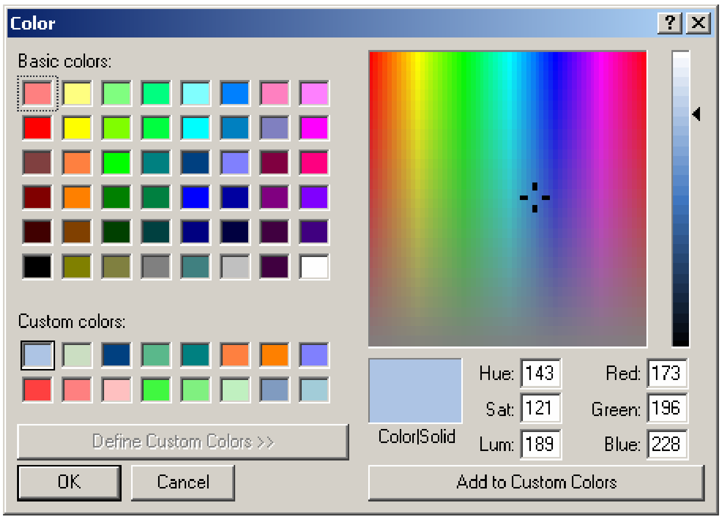
- Size (fraction of screen height): defines the size of the different markers in the GUI
- Tspoint size, default=0: defines the size of TSpoint markers (0=same than other markers)

Grid colors

- Grid water colour: user can define the colour of the water in the model window. To change the colour,
  a. click the box where the colour is shown and click **Select colour**



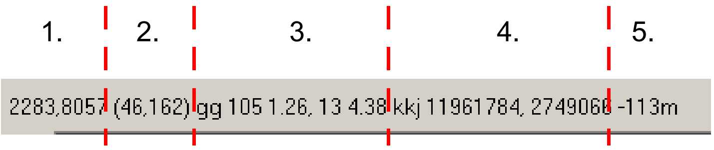
  b. select the wanted colour from the **Color** window and click **OK**



- Grid land colour: defines the colour of the land area in the model window
- Grid border colour: defines the colour of the grid box borders
- Grid shore show elevation: defines the elevation of shown water level in the model window

Coordinates

This section gives the user options to select which parameters are shown in the GUI. The information of the grid is shown on the next to the toolbar as shown in illustration below:

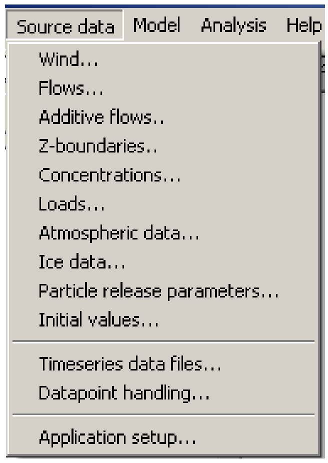


1. location of the pointer in the model grid in meters measured from the origin
2. grid box where pointer is (x, y)
3. location of the pointer in geographical coordinates
4. location of the pointer in utm coordinates
5. level of the grid box

Map

There is option to use background map in the model window. Browse the map location and select the coordinate system of the selected file. The map needs to be in *.dig format. Use RLGis (more about the RLGis can be found from **RLGis manual**) to convert the file from e.g. *.shp-file to *.dig-file.

7 SOURCE DATA



This chapter deals with the source data of the model, its handling and importing. Different source data types are introduced.

The chapter is divided to five parts:

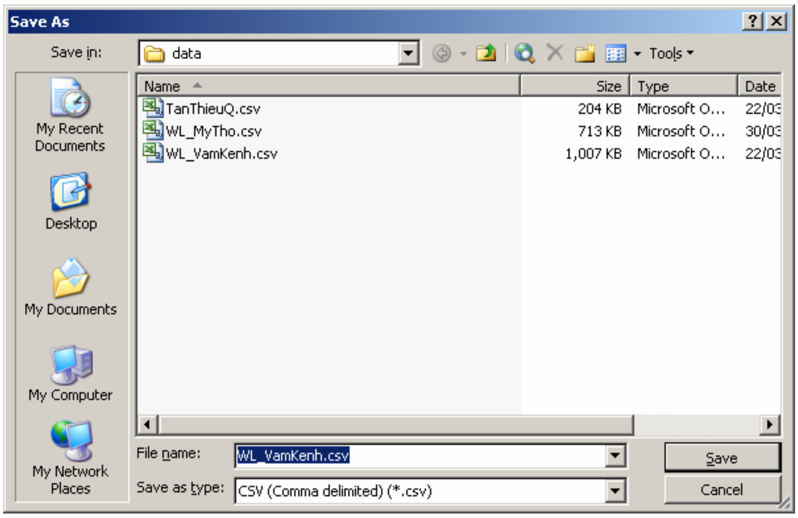
7.1 Importing timeseries
7.2 Boundary conditions
7.3 Timeseries data files
7.4 Datapoint handling
7.5 Application setup
7.6 Land use

7.1 IMPORTING TIMESERIES

The model needs timeseries data for boundary conditions for the model and to validate the model results. Normally the raw-data is in *.xls or *.txt format and need to be imported to the txd-format used by the EIA Model system. Follow the steps below to import timeseries data from different format to the model supported format.

At the moment the import of timeseries is working from both, “comma delimited” and “tab delimited” data formats. The data can be imported from txt-file and csv-file.

1. If the data is in **xls-format** it should be saved sheet by sheet to the csv-format. Follow the instructions below (if the data is in txt-format already you can go to step 2.
   a. Open the *.xls –file
   b. Select the sheet which includes the data you want to import to the model
   c. Select **File – Save as…**
   d. **Browse** for the location you want to save the file and edit the name of the file if needed
   e. Select CSV (Comma delimited) (*.csv) for the **Save as type**:



f. Click **Save**

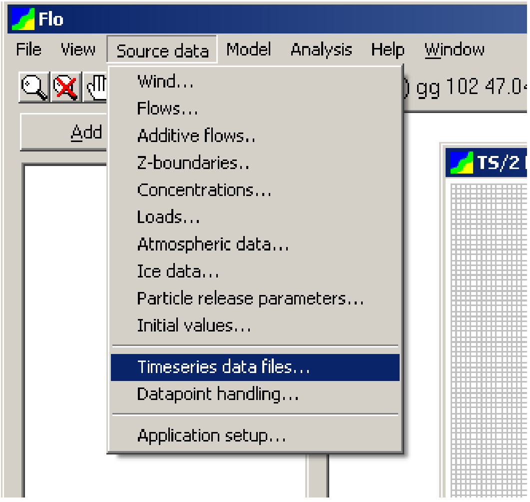
2. Now the data can be imported to the model required *.txd format. Example of the txd2 format is presented in Box 3. More about the txd-format can be read from [Section 3.2.3 - TXD file format](#).

Box 3. txd-data format example.

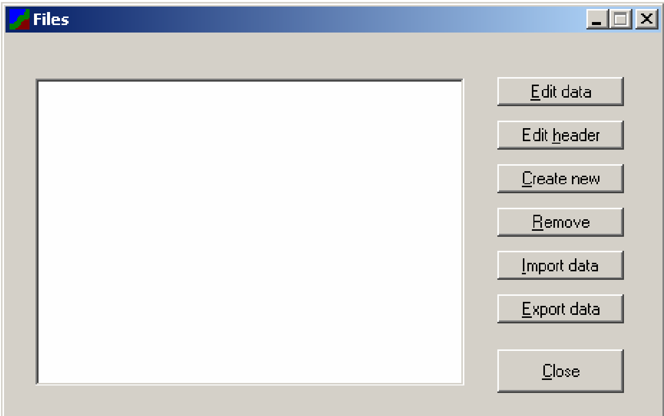
```
txd2
point PREKKDAM
xpos 478240
ypos 1305170
location "Prek Dam daily"
statid 1
time date "YYYYMMDD"
real HOUR_hh.h 5
real FLOW_m3/s 10
real WATL__m__ 10
data
19920101 24    -4533.09   4.16
19920102 24    -4461.74   4.09
19920103 24    -4378.95   4.01
19920104 24    -4326.55   3.96
19920105 24    -4284.25   3.92
19920106 24    -4230.91   3.87
19920107 24    -4166.23   3.81
19920108 24    -4111.77   3.76
19920109 24    -4089.83   3.74
19920110 24    -4045.72   3.7
```

a. Open the model application file (*.fld), if it is not already open, to which you want to insert the timeseries data

b. Select **Source data – Timeseries data files**

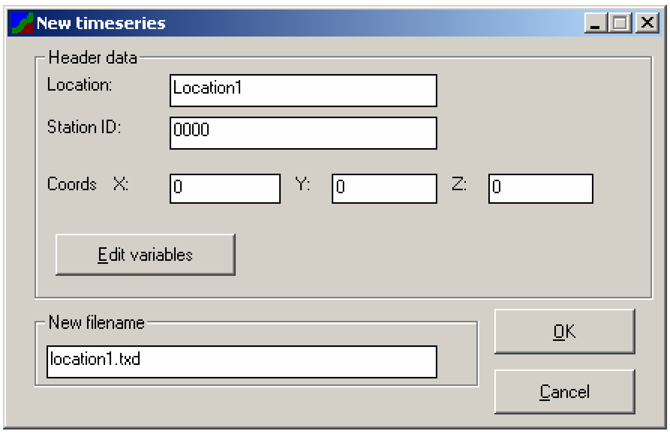


c. The following window appears:



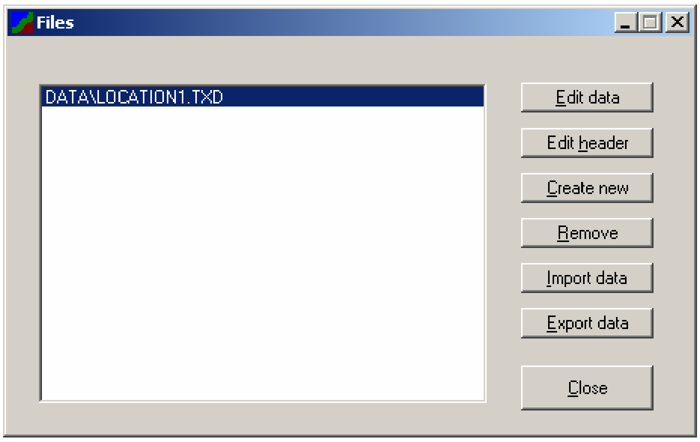
d. If there are no data files yet in your data-folder you have to first create new file before you can import data → press **Create new**

You have to create new data file if your Files-window is empty. After that you can proceed with Importing data.

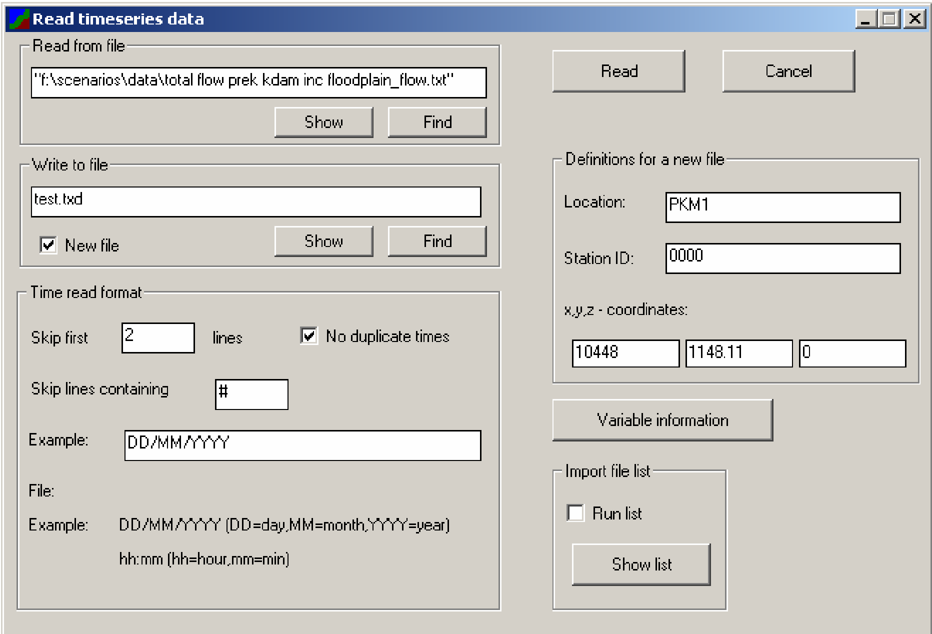


e. Press **Ok**

f. New data file has been created as shown below.



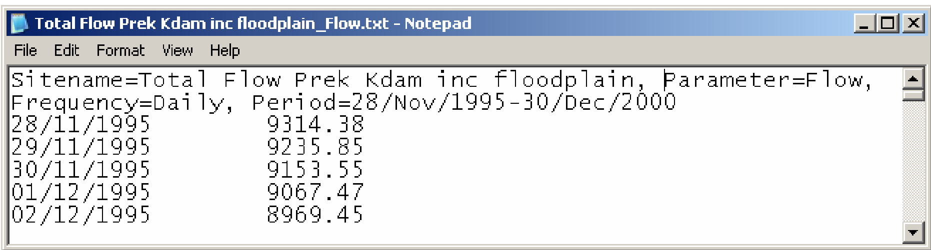
g. Now you can press **Import data** and following window will appear:



h. select the “Read from file” either by writing the name of the file or pressing **Find** from where you can browse the file from the right destination

i. write the name of the file in format (*.txd) to the “Write to file” field. The location is by default in model’s data-folder

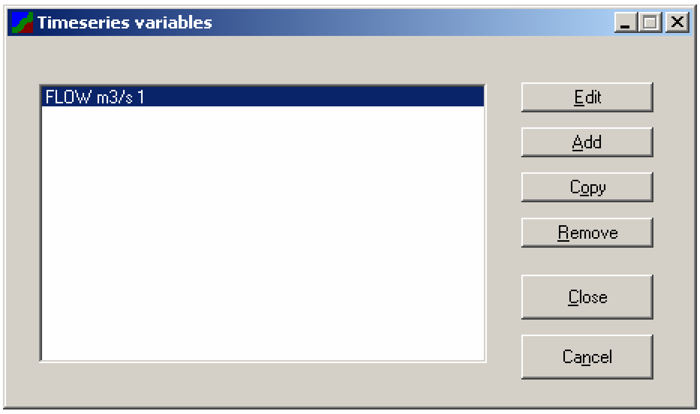
j. press **Show** under the “Read from file” field to see the file which is being imported to calculate the lines to be skip before the data itself start. This number of lines have to be write to the box on “Skip first □ lines”. On the following file there are two (2) lines to skip.



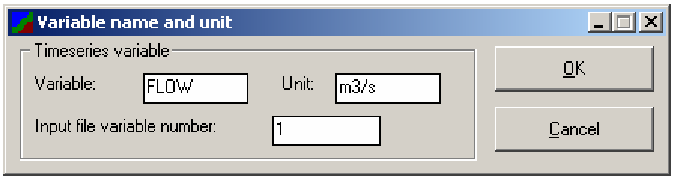
k. give the format of the date following the rules (note capitals in dates):
DD=day, **MM**=month, **YYYY**=year
hh=hour, **mm**=minute

l. After the time read format is given you have to fill the "Definitions for a new file" field where you give the location, station ID and x,y,z-coordinates.

m. last press **Variable information** to set up the variables to be read from the input file. Following window appears:



n. If the variable field is empty you have to **Add** new variable by pressing Add. If you want to edit existing variable press **Edit**

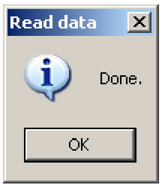


o. Select the name for the variable and unit. Then you have to look from the input file the order of the variable you are describing (e.g. if the first row of the data is flow data and second water level data, select 1 for the flow and 2 for the water level)

p. Press **OK**

q. When you have finished the variables input press **Close** in "Timeseries variables" window to return to the main Import window

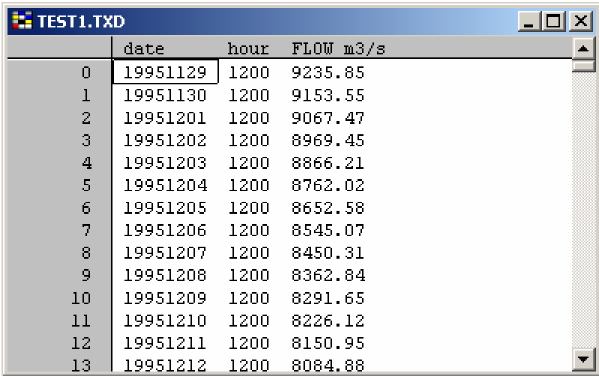
r. If all the information are correct, press **Read** and the program announced



If there occurs error in the data importing, check following:

- Variable information is correct
- Input file name
- Write to file field – you have to have the .txd after the file name
- Check the date and time format

3. you can see the data by pressing **Edit data** in "Files" window and the data is shown as follow



TEST1.TXD

	date	hour	FLOW m3/s
0	19951129	1200	9235.85
1	19951130	1200	9153.55
2	19951201	1200	9067.47
3	19951202	1200	8969.45
4	19951203	1200	8866.21
5	19951204	1200	8762.02
6	19951205	1200	8652.58
7	19951206	1200	8545.07
8	19951207	1200	8450.31
9	19951208	1200	8362.84
10	19951209	1200	8291.65
11	19951210	1200	8226.12
12	19951211	1200	8150.95
13	19951212	1200	8084.88

It is always good to check the imported data and that it corresponds to the original data, especially if there are multiple variables in original data.

4. When you have finished your data importing press **Close** in "Files" window and return to the main model window

7.2 BOUNDARY CONDITIONS

Boundary conditions give the input data to the model on its boundaries. The following boundary conditions exist in the EIA 3D model system:

- Wind
- Flows
- Additive flows
- Z-boundaries
- Concentrations
- Loads
- Atmospheric data
- Ice data
- Particle release parameters
- Initial values

7.2.1 Adding new boundary condition

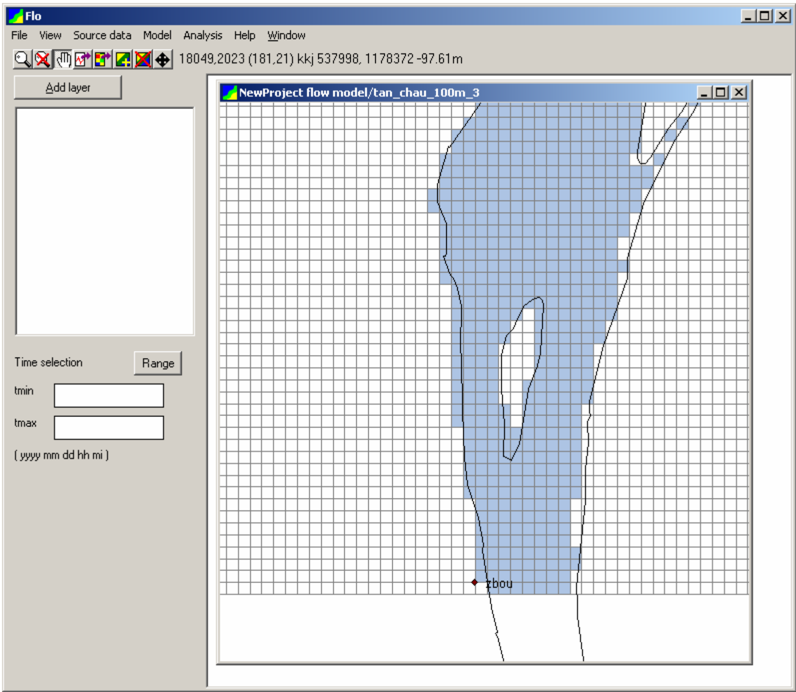
The user has two options to add new boundary condition or edit existing one to the model application:

- Through the model main window map interface

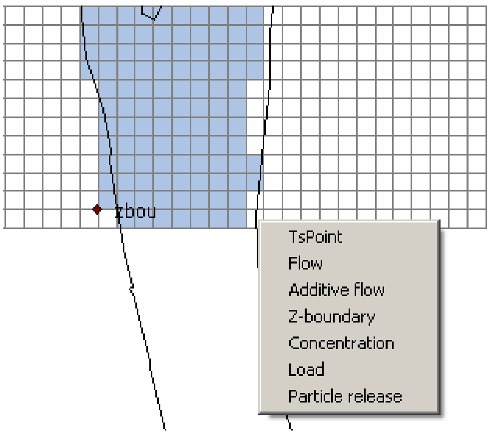
- Through the Source data menu

Adding new data through the map interface

1. check that the main model window is open
2. Zoom into the area you want to add the boundary condition



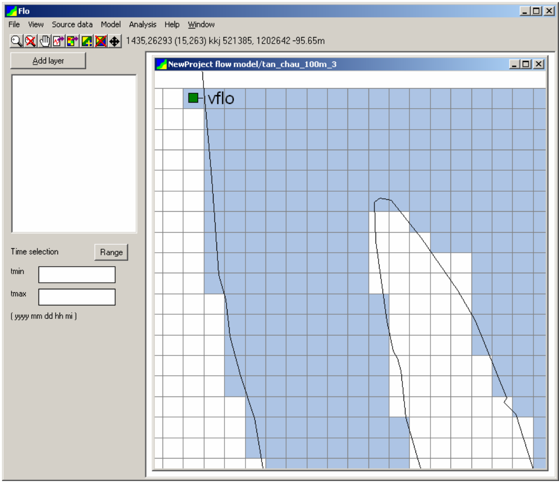
3. Select **Add item** tool  from the toolbar and keep left mouse button pressed down when selecting the new boundary condition place (drag the line through the boundary) and the menu appears on the screen



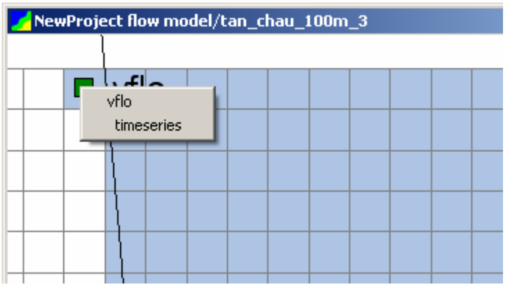
4. Select the boundary condition you want to add with the left mouse button and window for that boundary condition parameter set-up will appear. See the following sections for more detailed information for each boundary condition.

Edit existing data through the map interface

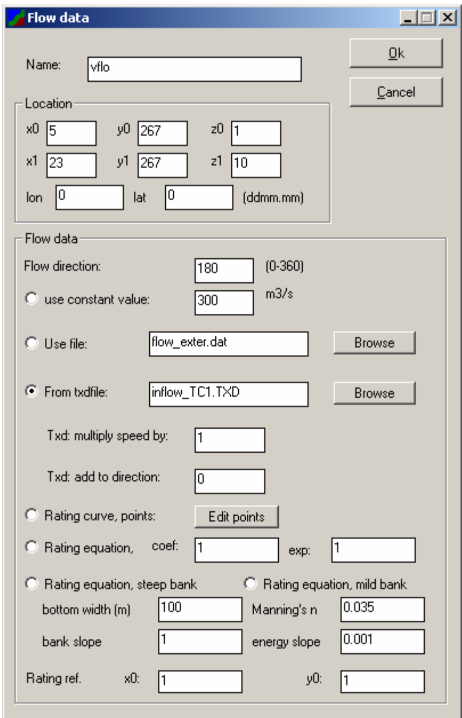
1. In the main model window zoom into the area where the boundary condition you want to edit is



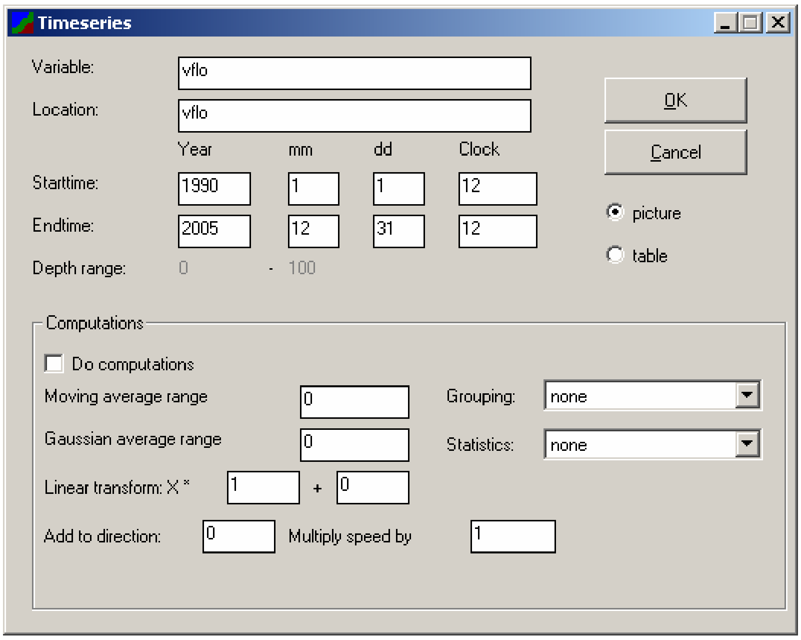
2. release the zoom by clicking the right mouse button
3. click the boundary condition data point with the left mouse button and following menu appears



4. To edit the header information, click the **name** of the boundary condition (this case **vflo**) and the boundary condition parameter window appears. More about the options for each boundary condition, see the following sections where the settings fro each boundary condition have been explained in details.



5. To see and/or edit the boundary condition data, select **timeseries** and the Timeseries window appears



a. First user has to define whether the data will be shown in table or picture format
b. Then user has to define the start and end date of the data that will be shown
c. User is also able to do to simple computations. The Do computations check box needs to be checked to make the computation options active.
d. Click **OK** to do the selected action.

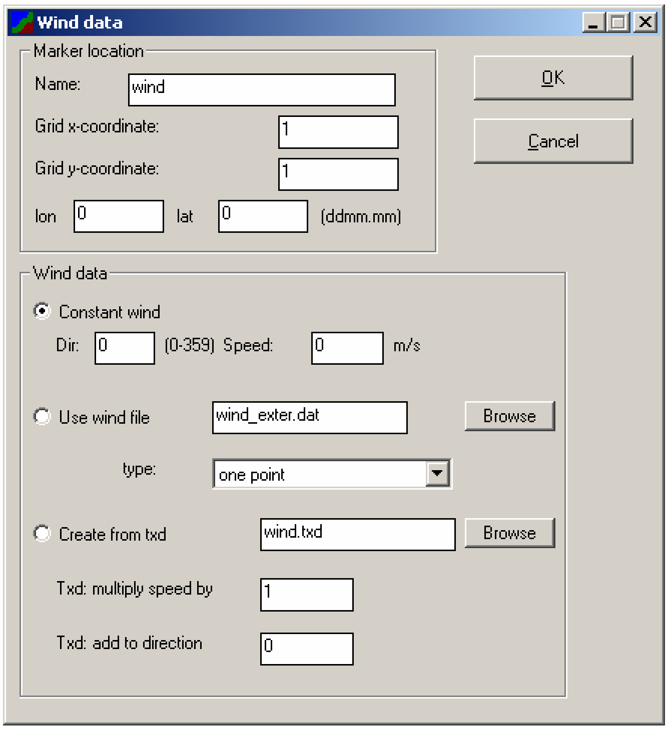
Adding new data through the Source data menu

This option is presented separately to each boundary condition in the following sections.

7.2.2 Wind

Variable name:	WSPD	wind speed
	WDIR	wind direction
Unit:	[m/s]	wind speed
	[degree]	direction from where the wind is blowing

The following window appears when you click the **Source data – Wind...** You can access to the wind data also by clicking the Wind symbol in the model window with left mouse button and select wind (or if other, the name of the symbol).



Name - Symbol title (only for display)

Grid x, y-coordinates - Coordinates in grid (used only for display)

Constant wind - checked if constant wind used
Direction - incoming wind direction in degrees from north, north=0, east=90
Speed - wind speed in m/s

Use wind file – Checked if a file is provided with specific meteorological format. Use Browse to locate the wind data file.

Type:
constant = constant wind
one point = wind time series from one point
areas: coded data = matrix of wind data using meteorological observation coding
areas: u- and v-components = wind u- and v-componentes given separately
areas: dir, speed = wind given as direction and speed
EXV = flow model format
stress (ETA) = wind stress file from ETA meteorologial model
Areas can cover range of grid cells or each grid cell separately.

Create from txd - Use wind data from a timeseries file. Use browse to locate the timeseries file.
txd: Multiply speed by - Multiply the wind speed in the txd-file with this number, applies only when create from txd option is selected
txd: Add to direction - Add this number to the wind direction in the txd-file, applies only when create from txd option is selected

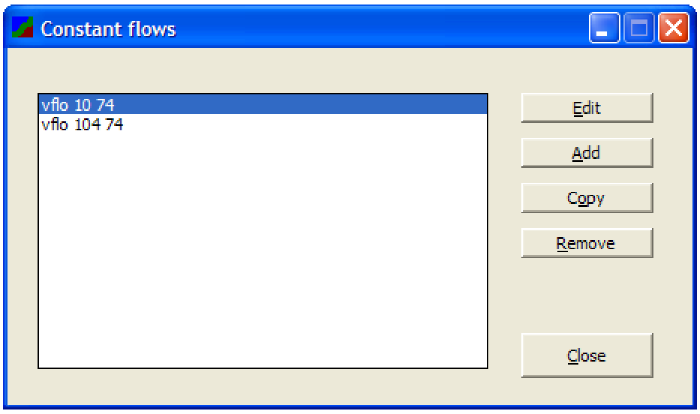
Wind is the roughly horizontal movement of air (as opposed to an convective air currents) caused by uneven heating of the Earth's surface. It occurs at all scales, from local breezes generated by heating of land surfaces and lasting tens of minutes to global winds resulting from solar heating of the Earth. The two major influences on the atmospheric circulation are the differential heating between the equator and the poles, and the rotation of the planet (Coriolis effect).

[Source: Wikipedia (www.wikipedia.org)]

7.2.3 Flow

Variable name: FLOW
Unit: [m^3/s]

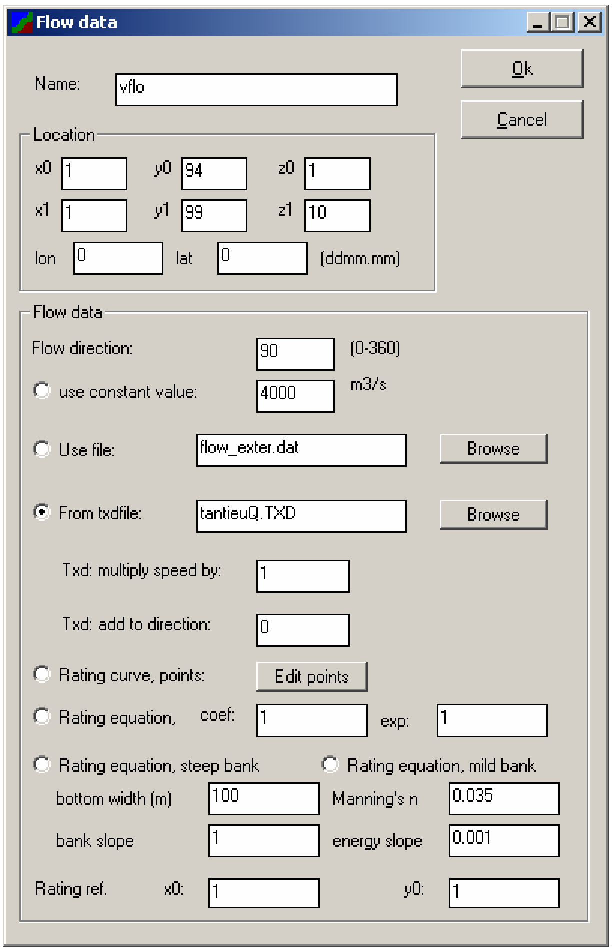
Flow defines rivers and boundary flows used by the model. Select **Source data – Flows…** to open the list of the *Flows* as presented below:



List actions:

Edit - edit item data
Add - add new item
Copy - create a new item by copying an existing item
Remove - remove an item
Close - close the list window

The Flows window (see below) will be opened when either existing *Flow* item is edited (**Edit**) or new *Flow* item is created (**Add**)



Flow data:

Name - symbol name

Location

x0,y0,z0 - grid area bottom left surface-layer corner coordinates

x1,y1,z1 - grid area top right bottom-layer corner coordinates

lon, lat – coordinates in lat-lon system (not needed if UTM system is used)

Coordinates may contain land grid boxes but may not be greater than grid dimensions.

Constant flow

Flow direction - incoming flow direction in degrees from north, north=0, east=90 (see Figure 12).

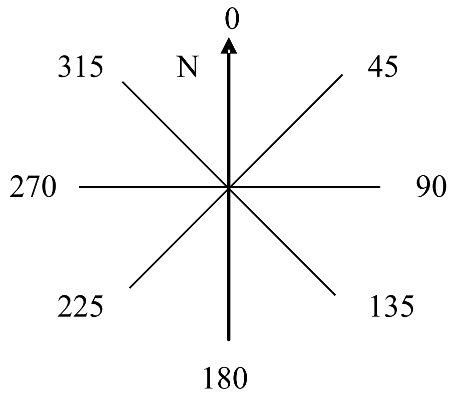


Figure 12. Flow directions in EIA 3D model when grid x-axis direction is set to be 90 (see Section 8.1.3 - Grid x-axis direction).

Use constant value – if checked the constant flow is used

There must be value greater than 0 in the *Use constant value* data window even if the txd-file or data-file is used. If the constant flow is 0, the flow is not used for computation.

Use file - If checked use the file defined in data file box

From txdfile – if checked the timeseries txd-file defined in the data file box

Txd: multiply file values by - multiply timeseries values with this value

Txd: add to file values - add this value to timeseries data

Rating curve, points – if selected, give water surface elevations and corresponding discharges by pushing Edit points-button

Rating curve, equation – $Q = a * z^c$ where Q is discharge and *z* water elevation; this option may require that water elevation is adjusted in the model code for other than absolute reference system or a reference system option is added to the user interface and code; the same discussion applies to the options below also

coef: *a*

exp: *c*

Rating equation, steep bank – $Q = \frac{z^{5/3} \cdot (b + z / S) \cdot \sqrt{S_0}}{M}$

Rating equation, mild bank – $Q = \frac{(b \cdot z + z^2 / S)^{5/3} \cdot \sqrt{S_0}}{M \cdot (b + 2 \cdot z \cdot \sqrt{1 + 1 / S^2})}$

Manning's n: *M*

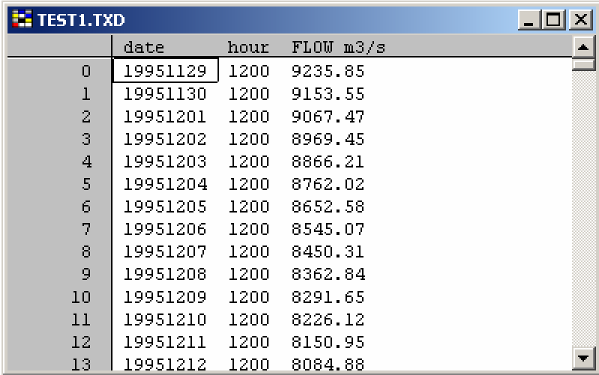
Energy slope: S_0

Bottom width: *b*

Bank slope: *S*

Rating ref – the reference cell from the where the water level is taken for the rating curve (ideally the location where the water level is measured when the rating curve has been created)

If the rating curve doesn't work properly, check that the reference point is correctly defined.



	date	hour	FLOW m3/s
0	19951129	1200	9235.85
1	19951130	1200	9153.55
2	19951201	1200	9067.47
3	19951202	1200	8969.45
4	19951203	1200	8866.21
5	19951204	1200	8762.02
6	19951205	1200	8652.58
7	19951206	1200	8545.07
8	19951207	1200	8450.31
9	19951208	1200	8362.84
10	19951209	1200	8291.65
11	19951210	1200	8226.12
12	19951211	1200	8150.95
13	19951212	1200	8084.88

Figure 13. Example of the flow data in the timeseries table.

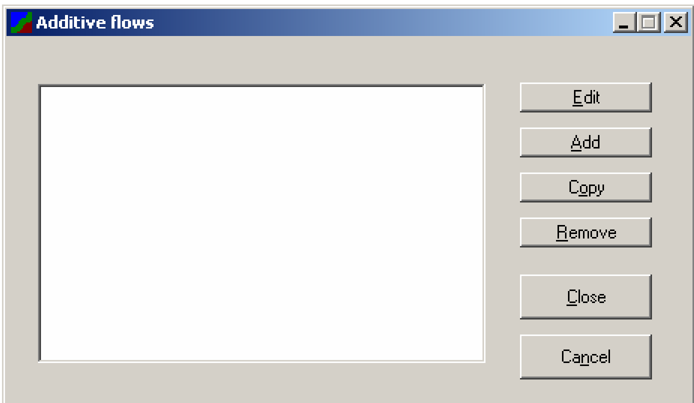
Discharge (=flow): In hydrology, the discharge of a river is the volume of water transported by it in a certain amount of time. The unit used is usually m^3/s (cubic meters per second). For example, the average discharge of the Rhine river is 2200 m^3/s. The greater the discharge of a river, the more ability it has to carry sediment. The discharge of a river can be estimated by taking the area of a cross-section of the river and multiplying it by the river's average velocity.

[Source: Wikipedia (www.wikipedia.org)]

7.2.4 Additive flow

Variable name: FLOW
Unit: [m^3/s]

Defines additive flows such as exhaust pipes. Similar to flow except that additive flow adds water to a grid area while constant flow sets the flow on the area to a given value. Select **Source data – Additive flows…** to open the list of the *Additive flows* as presented below:



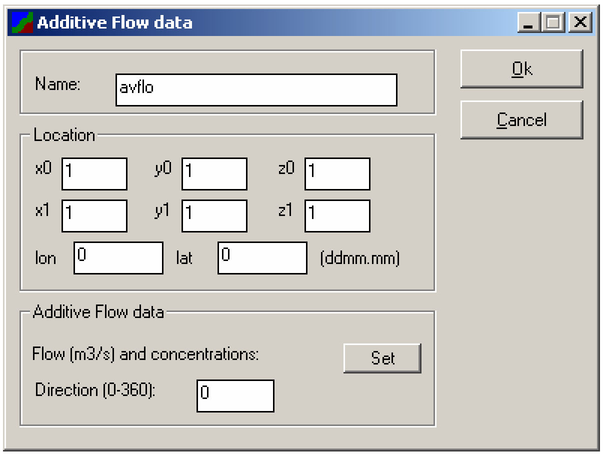
List actions:

Edit - edit item data

Add - add new item

Copy - create a new item by copying an existing item
Remove - remove an item
Close - close the list window
Cancel - close the list window

The Additive flows window (see below) will be opened when either existing *Additive flow* item is edited (**Edit**) or new *Additive flow* item is created (**Add**)



Additive flow data:

Name - symbol name

Location

x0,y0,z0 - grid area bottom left surface-layer corner coordinates

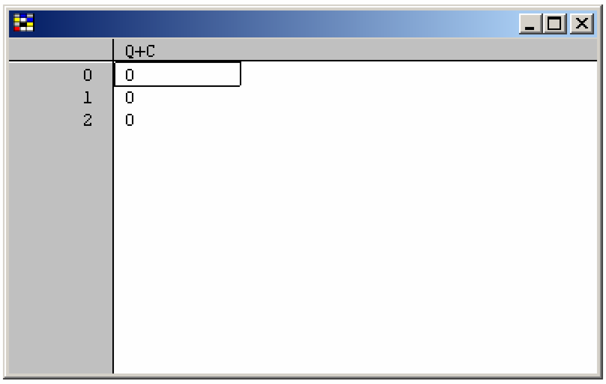
x1,y1,z1 - grid area top right bottom-layer corner coordinates

lon, lat – coordinates in lat-lon system (not needed if UTM system is used)

Coordinates may contain land grid boxes but may not be greater than grid dimensions.

Additive flow

Flow (m3/s) and concentration – selecting **Set** pops up a matrix, the first line of which contains the flow amount in m3/s and the rest of the lines each value for each density variable in the additive flow.

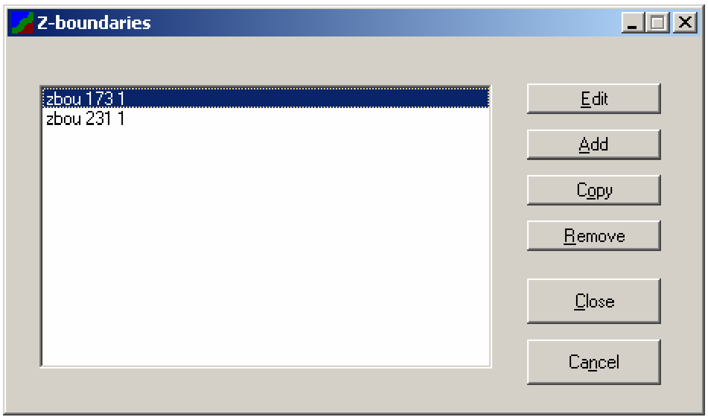


Direction - incoming flow direction in degrees from north, north=0, east=90, etc (see Figure 12).

7.2.5 Z-boundary

Variable name: WLEV
Unit: [m] (normally above mean sea level or some other reference level)

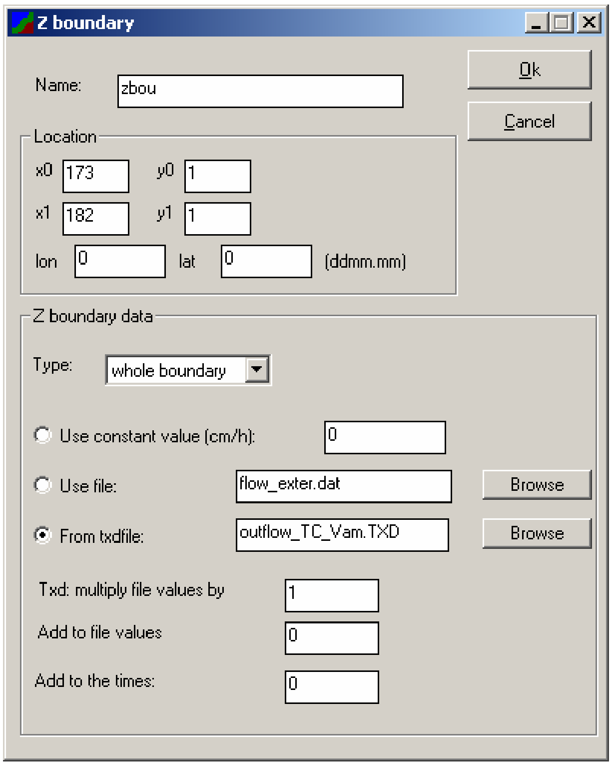
With Z-boundary option the user is able to set the boundary water level to the wanted level e.g. downstream of the river model application, tidal water level in shore application, etc. Select Source data – Z-boundaries… from the main menu to open the list of the concentrations as presented below:



List actions:

Edit - edit item data
Add - add new item
Copy - create a new item by copying an existing item
Remove - remove an item
Close - close the list window
Cancel - close the list window

The **Z Boundary** window (see below) will be opened when either existing Concentration item is edited (**Edit**) or new Concentration item is created (**Add**)



Name - Symbol name

Location

x0, y0 - grid area bottom left surface-layer corner coordinates

x1, y1 - grid area top right bottom-layer corner coordinates

lon, lat – coordinates in lat-lon system (not needed if UTM system is used)

> Coordinates may contain land grid boxes but may not be greater than grid dimensions.

Z boundary data

Type – the concentration variable to set

Use constant value (cm/h) – if checked the constant increase/decrease of water level is used (negative value is decreasing water level, 0 keeps the water level stagnant)

Use file – if dat-file is used for the boundary condition, check the check box and browse the file

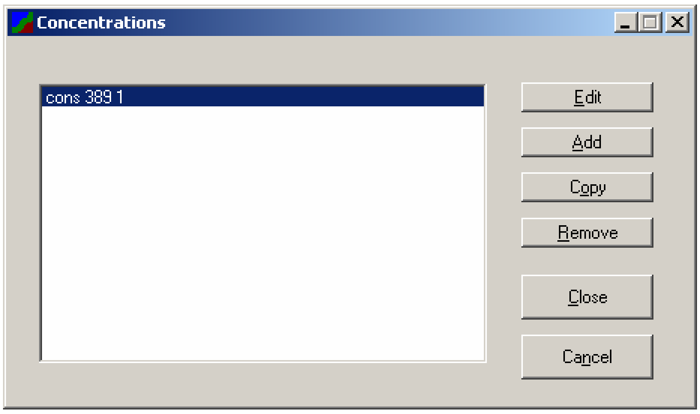
From txdfile – if checked the timeseries txd-file defined in the data file box will be used to define the Z boundary level

Txd: multiply file values by – multiply timeseries values with this value
Add to file values – add this value to timeseries data
Add to the times – add this value to the time data

7.2.6 Concentration

Variable name: Varies (see Section 3.2.3 - TXD file format for the variable names)
Unit: [mg/l] for substance or [C] for temperature

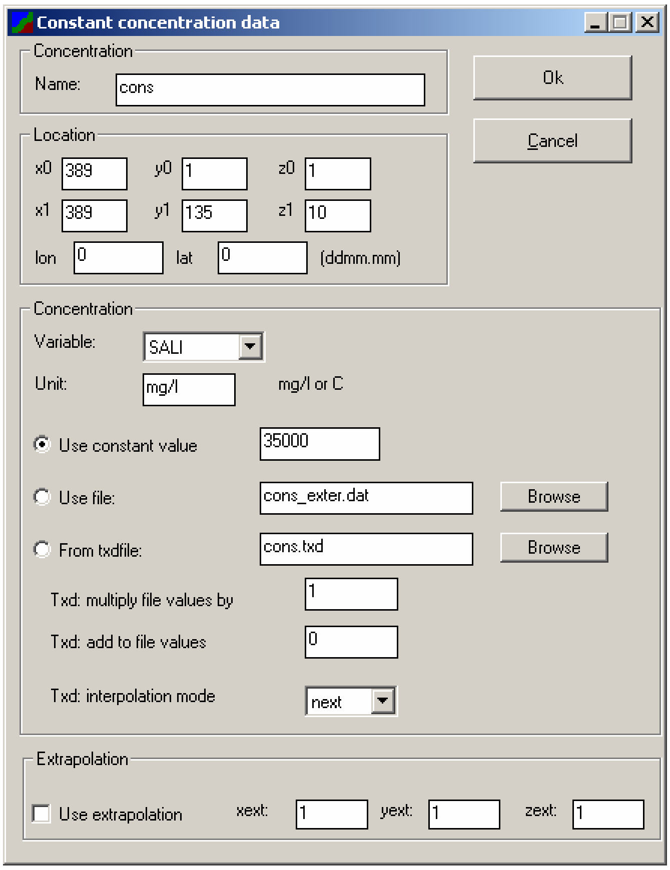
List of boundary/constant concentrations. There must be a boundary condition for each computed variable at each flow border location. Constant concentration sets the concentration of a given variable to be the given value at given location at all times. Select **Source data – Concentrations…** to open the list of the concentrations as presented below:



List actions:

Edit - edit item data
Add - add new item
Copy - create a new item by copying an existing item
Remove - remove an item
Close - close the list window
Cancel - close the list window

The Concentration data window (see below) will be opened when either existing Concentration item is edited (**Edit**) or new Concentration item is created (**Add**)



Concentration data:

Name - Symbol name

Location

x0,y0,z0 - grid area bottom left surface-layer corner coordinates

x1,y1,z1 - grid area top right bottom-layer corner coordinates

lon, lat – coordinates in lat-lon system (not needed if UTM system is used)

Coordinates may contain land grid boxes but may not be greater than grid dimensions.

Concentration

Variable - the concentration variable to set

Unit – either mg/l for substance or C for temperature

Use constant value – if checked the constant load is used (negative value is residency time)

Use file - If checked use the file defined in data file box

From txdfile – if checked the timeseries txd-file defined in the data file box

Txd: multiply file values by - multiply timeseries values with this value

Txd: add to file values - add this value to timeseries data

Txd: interpolation mode – specifies how concentrations are interpolated in between given values:

- Linear: linear interpolation
- Previous: nearest previous value is used
- Next: nearest next value is used

Use extrapolation – values are extrapolated to the boundary from a point which is defined by coordinates (xext, yext, zext).

Concentration is the measure of how much of a given substance there is mixed with another substance. This can apply to any sort of chemical mixture, but most frequently is used in relation to solutions, where it refers to the amount of solute dissolved in a solvent.

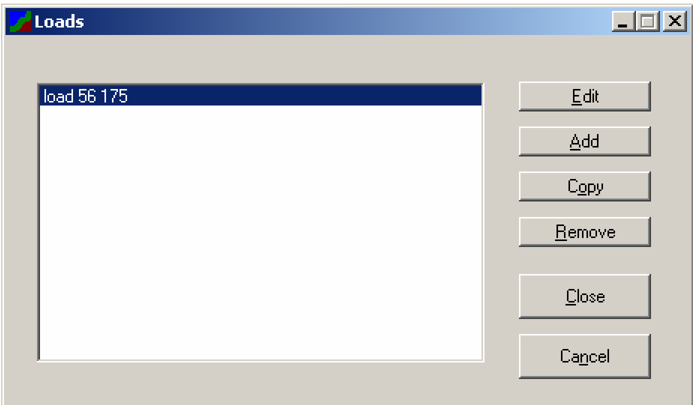
Unit: unit is either [mg/l] (= [g/m^3]) for substance (e.g. total suspended solids, oxygen) and [C] for temperature

[Source: Wikipedia (www.wikipedia.org)]

7.2.7 Load

Variable name: Varies (see Section 3.2.3 - TXD file format for the variable names)
Unit: [mg/l] for substance or [C] for temperature

List of point loadings. Each point load item handles one variable at one location. If there are releases of several nutrients you must add one point load for each variable. Select **Source data – Loads…** to open the list of the loads as presented below:



List actions:

Edit - edit item data

Add - add new item

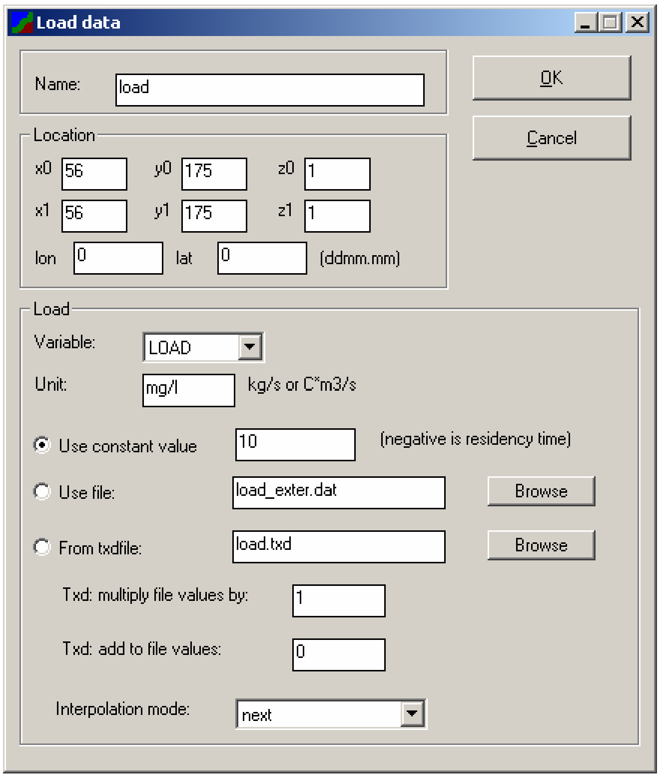
Copy - create a new item by copying an existing item

Remove - remove an item

Close - close the list window

Cancel - close the list window

The Load data window (see below) will be opened when either existing load item is edited (**Edit**) or new load item is created (**Add**)



Load data:

Name - symbol title

Location

x0,y0,z0 - grid area bottom left surface-layer corner coordinates

x1,y1,z1 - grid area top right bottom-layer corner coordinates

lon, lat – coordinates in lat-lon system (not needed if UTM system is used)

Coordinates may contain land grid boxes but may not be greater than grid dimensions.

Load

Variable - the concentration variable to set

Unit – unit of the load: [kg/s] if the variable is a substance and [$C*m^3/s$] if the variable is temperature

Use constant value – if checked the constant load is used (negative value is residency time)

Use file - If checked use the file defined in data file box

From txdfile – if checked the timeseries txd-file defined in the data file box

Txd: multiply file values by - multiply timeseries values with this value

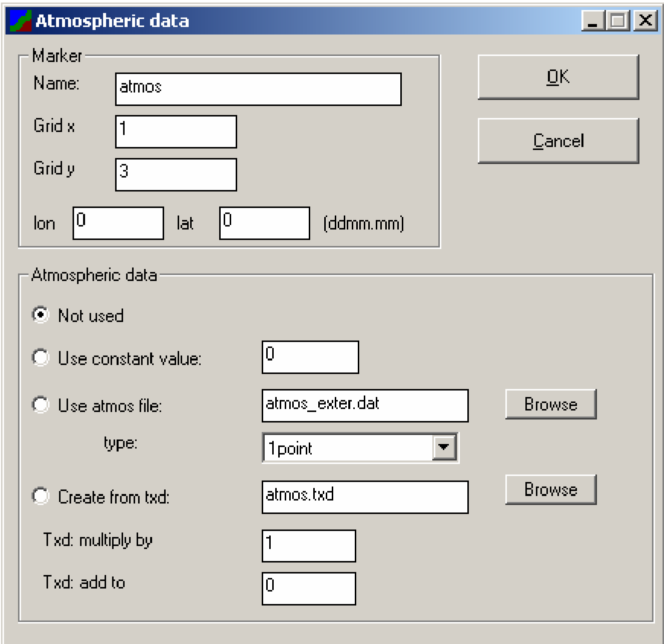
Txd: add to file values - add this value to timeseries data

Load is the measure of how much of a given substance there is released to the water in relation to the time.

Unit: the unit is [kg/s] for substance and [C*m^3/s] for temperature

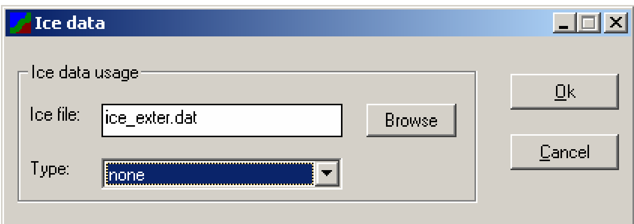
7.2.8 Atmospheric data

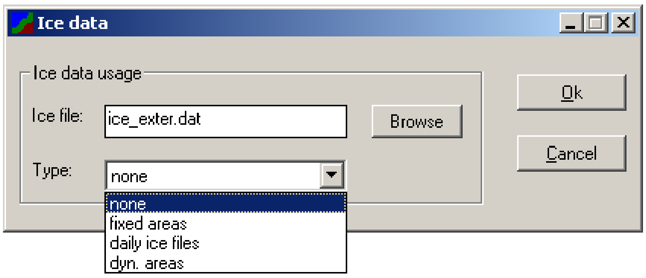
Atmospheric data is required for water temperature simulation. Heat flux simulation uses either measured radiations or calculates radiation from atmospheric data including air temperature, humidity, and pressure and cloudiness. Also precipitation is given in this dialog when it is included in an application. Atmospheric data can be given homogenously for whole calculation area or specified separately for sub-areas or every grid cell. At the moment this option is not used in the Mekong applications.



7.2.9 Ice data

Model can calculate ice or ice data can be given from a file. Not applicable to Mekong.



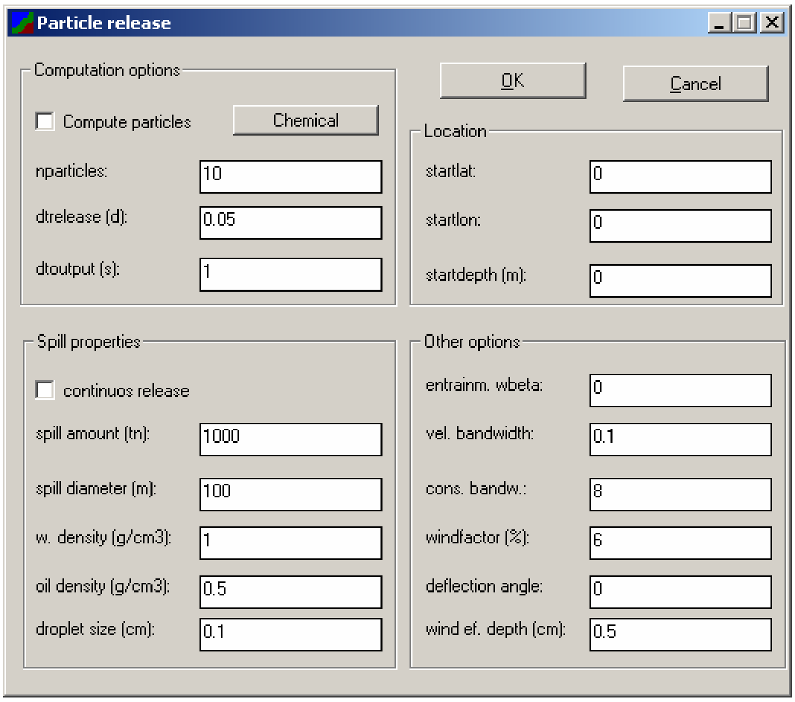


Different types of ice data are:

- Fixed areas
- Daily ice files
- Dyn. areas

7.2.10 Particle release parameters

User is able to model particle movements in the model application. By selecting **Source data – Particle release parameters…** the user is able to activate the particle computation and define detailed parameters for the particles released as presented below:



Computation options

Chemical – chemical reactions and physical processes can be specified to be calculated; requires separate chemical parameter file; includes evaporation, emulsification, dissolution and decay; processes are described differently on the surface, water column and bottom

nparticles - total number of particles used in the simulation or number of particles released each time step in case of continuous spill; oil, chemicals, fish larvae and floating objects such as boats are calculated as particles (Lagrancian system) for accurate transport without numerical diffusion

dtrelease - time step in days between releases (see Spill properties) dtoutput - output timestep for numerical and graphical spill characteristic output

Spill properties

continuous release – if this is not checked, the spill happens in the beginning of the simulation; if this checked substance is spilled with dtrelease timestep interval

spill amount and diameter – initial spill mass and diameter

w. and oil density - water and oil densities

droplet size - droplet size in vertical dispersion

Location

startlat and startlon – initial spill coordinates in lat-long

startdepth - initial spill depth

Other options

entrainment w_β – vertical particle diffusion bandwidth; the vertical random motion component is $P(-1,1)\,W\sqrt{(6\frac{w_\beta}{\Delta t})}$, where P(-1,1) is random function with value between -1 and 1 and W is wind speed. To get actual displacement this term is multiplied by timestep Δt.

vel. bandwidth – horisontal particle probabilistic diffusion coefficient for particle velocity dependent component: $P(-1,1)\,u\,w_u$

cons. bandwidth – constant horisontal particle diffusion bandwidth

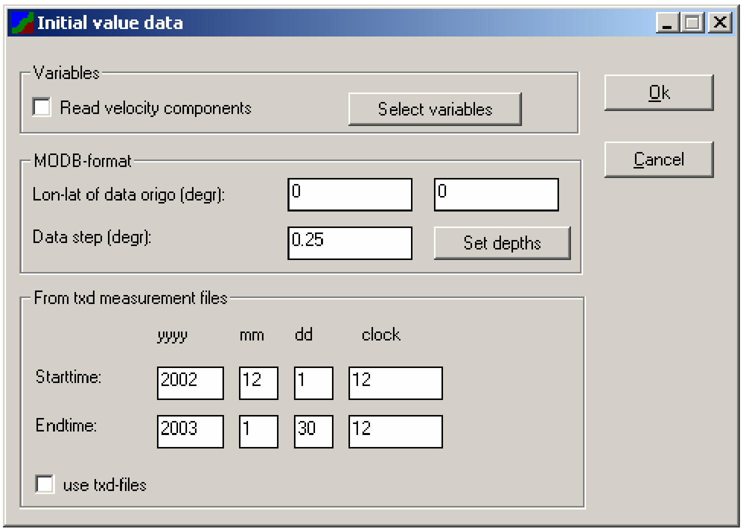
windfactor – percentace of wind affecting directly substance surface drift (if surface layer is thin, this can be 0)

deflection angle – wind impact deflection compared to the wind direction

wind ef. depth – effective depth of direct wind impact

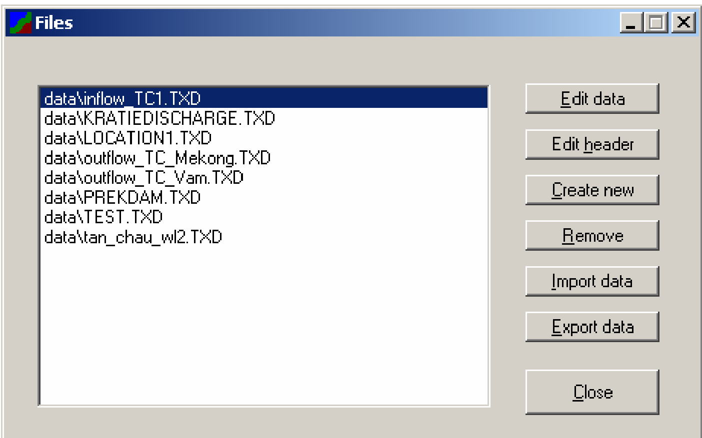
7.2.11 Initial value data

This option enables detailed setup of initial values. Used mostly in large sea areas where initial state has long impact. Not used in Mekong. See about setting of initial values below.



7.3 Timeseries data files

The timeseries data files, i.e. all txd files in data directory (see [Section 7.5 – Application setup]) can be seen, edited and exported in Timeseries data files window. To open the window, select **Source data – Timeseries data files…** from the main menu.



List actions:

Edit data - edit item data

Edit header – edit item's header

Create new – creates new timeseries data file

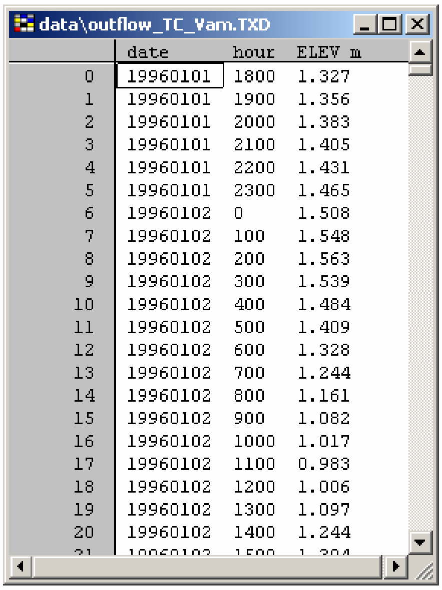
Remove – removes selected timeseries data file

Import data – import timeseries data from other file

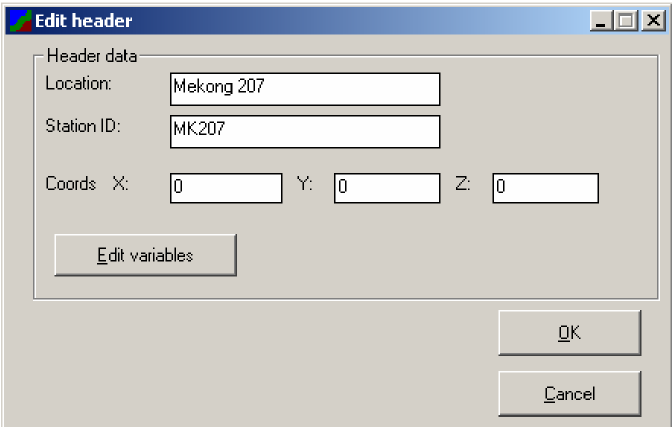
Export data – export timeseries data to txt file

Close – closes the window

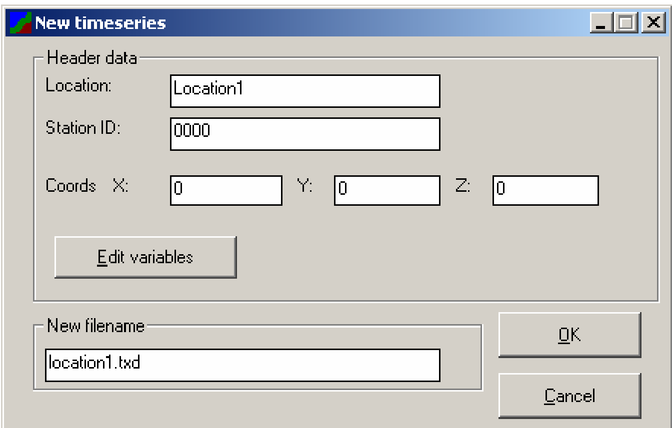
Edit data – Edit data command opens the data table which user can edit or make basic analysis if needed.



Edit header – with this command the user is able to update the header information (location, station ID, coordinates, and variables) of the timeseries data file.



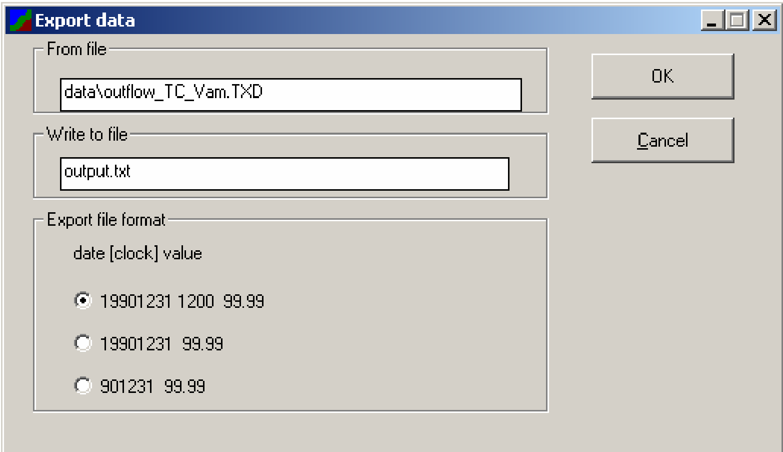
Create new – with this command the user is able to create a new timeseries data file as presented below.



Remove – the user can remove unneeded timeseries data files from the data directory by selecting the file and then pressing **Remove**

Import data – with this command user can import timeseries data to txd-format from other data formats. This is explained in more details in Section 7.1 – Importing timeseries.

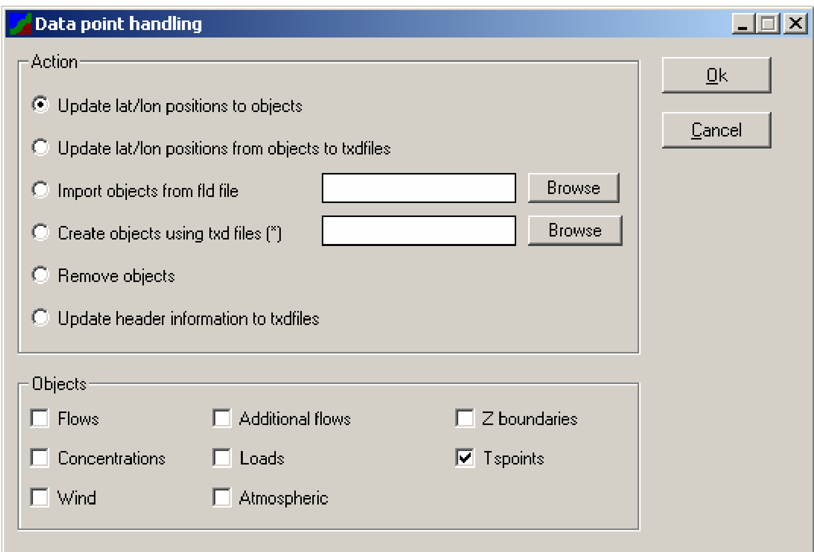
Export data – user is able to export the selected txd-file to txt-file with this command. The window for settings is presented below



7.4 Datapoint handling

Within the Data point handling window the following actions can be taken:

- Update lat/lon positions to objects
- Update lat/lon positions from objects to txd-files
- Import objects from fld file: imports selected objects from the other model application
- Create objects using txd files (*): the selected object can be done directly from the txd-file by using this action
- Remove objects: user can remove all the selected objects at the same time from the model application if needed
- Update header information to txd-files: updates the header information to txd-files



Follow the steps below to do required action:

1. select the object to where the action will be impacted under the **Objects** part of the window
2. select the action un **Action** part of the window
3. press **OK** to do the action

7.5 Application setup

This dialog allows setting model title, directories and computational model files. To open the application setup window select **Source data – Application setup…**

Title:

Project title - project title is displayed in the model window title.

Directories:

Flow directory - location in which the computations are and where computation results are stored. The directory should exist. Conventionally 'wq\' is used

Data directory - location in which the timeseries files (txd files) are stored. Conventionally 'data\' is used

Use relative paths – click on the "Use relative paths" if Flow and Data directories are in the same folder than your model file (*.fld). If those are somewhere else, click the "Use relative paths" off and give the complete path of the directories (e.g. C:\model\directories\)

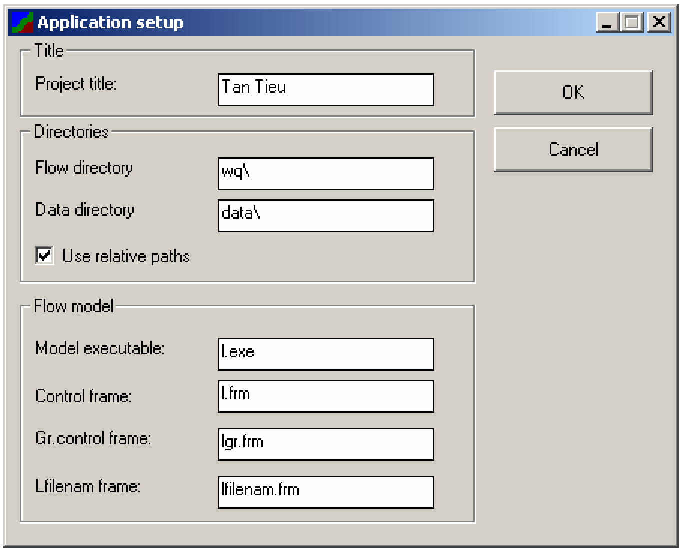
Flow model:

Model executable - name of the computational model program in relation to flow directory (if the l.exe file is not in the same directory than the model file (*.fld) give the path as well, in relative to the flow directory)

Control frame - name of the computation control file frame

Gr.control frame - name of the graphics control file frame

Lfilenam frame - name of the file list file frame

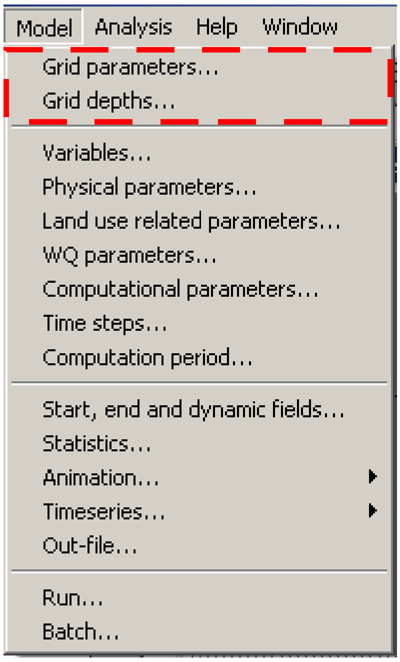


7.6 Land use

The model is using land use map to describe the density, height and oxygen consumption of vegetation. The model is using that information to calculate the vegetation impact on the wind, flow friction, and oxygen levels. The land use map needs to be first imported to the model application prior to set-up the parameters for each landuse class.

For more about the Landuse in the model see Section 9.3 - Landuse parameters.

8 MODEL GRID SETTINGS



This chapter deals with the model grid and settings related to it. It goes through the setting up of grid parameters and how the grid depths can be edited in the GUI.

The chapter is divided to two parts:

8.1 Grid parameters

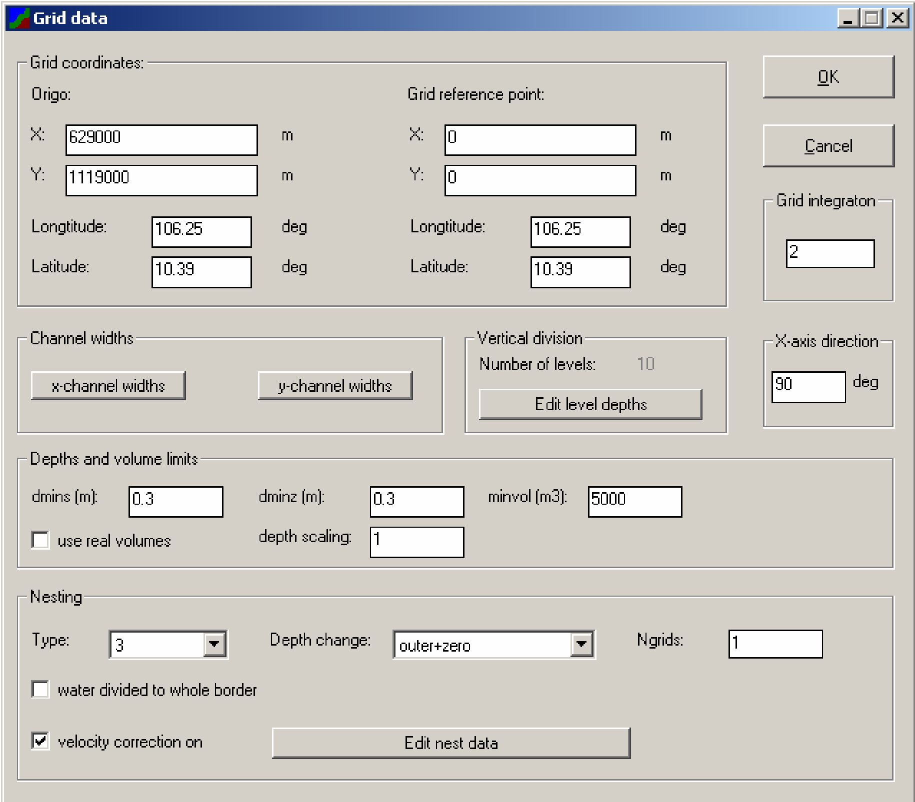
8.2 Grid depths

8.1 Grid parameters

In Grid parameters window the following grid parameters can be set:

- Coordinates
- Grid integration
- Grid X-axis direction
- Channel widths
- Vertical division
- Depths and volume limits
- Nesting
- Channels

The Grid parameter window is presented below, you can open it by selecting **Model – Grid parameters…**. In following sections each parameter is explained in more details with the instructions how to set up each one for the model.



8.1.1 Coordinates

In Grid coordinates sub-section of the Grid data window the user is able to define the coordinates of the model application origo in UTM and lon/lat systems. Both are needed for different purposes.

- UTM coordinates to locate the possible Timeseries data points and maps if they are in UTM projection
- Lon/lat coordinates to calculate the coriolis force and to locate spatial information and data (maps, TS points) which is possibly in lon/lat system.

Origo X, Y - Grid bottom left coordinates in UTM system

Grid reference point X, Y – reference point for the application's grid coordinates. Normally best to have X: 0 and Y: 0 when the grid coordinates are shown as meters from the origo.

Longitude - grid origo longitude coordinate

Latitude - grid origo latitude coordinate

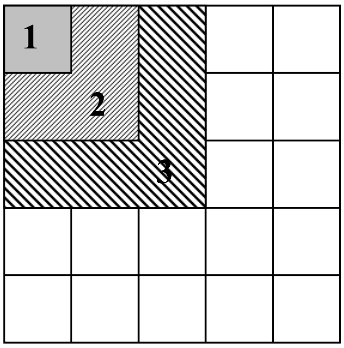
Model uses the Longitude and Latitude information to define the coriolis coefficient. Thus, it is important that you have right latitude and longitude coordinates in the model.

More about the coriolis: see box under the Section 9.2.5 – Miscellaneous where the coriolis force is explained and Chapter 21 – Mathematical Description of Water Flow where the mathematical description of the coriolis force is presented.

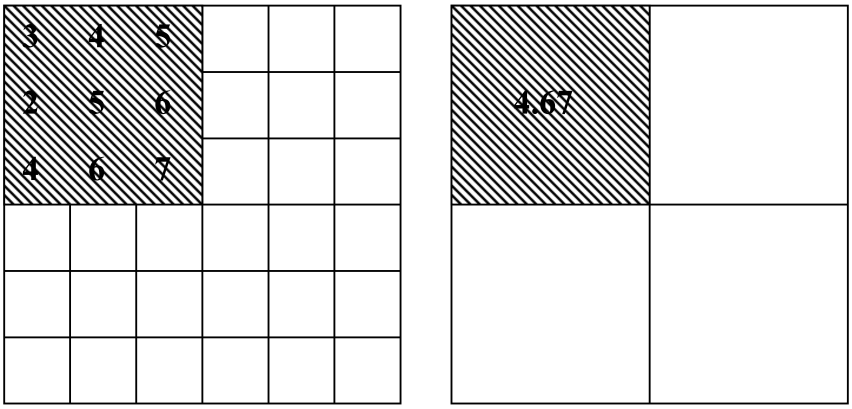
8.1.2 Grid integration

User is able to combine grid cells in order to make the computation faster. This option must be used cautiously, because for instance control structures will not necessarily be sensibly defined in an integrated grid. Thus grid integration is suitable for cases with no or simple control structures.

Grid integration idea is presented in figure below. The default is that the grid integration is 1, which means that the model is using the grid size defined by user when creating the grid (e.g. 100 m). If the grid integration is selected to be 2, then the size of one modelled grid cell becomes 2*original grid cell (in this case 200 m). For grid integration = 3 the size of one grid cell becomes 3. Below the principles are presented for how the elevation and land use information are calculated for the new integrated grid cell.



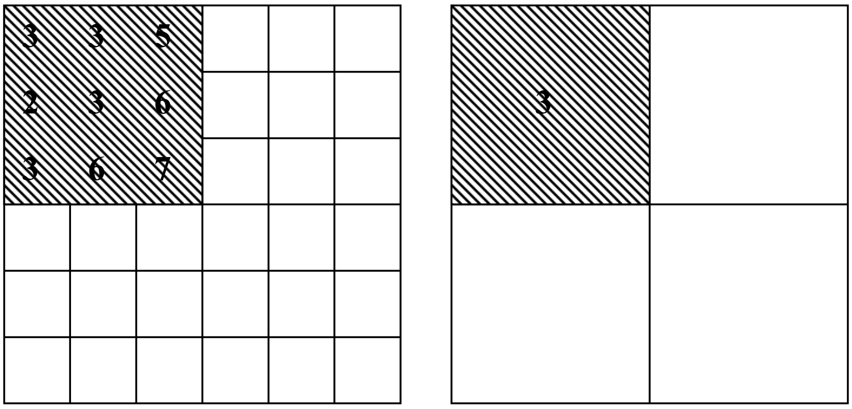
Elevation: the model calculates the average elevation value for integrated grid cells based on the original grid data.



Original elevation

New elevation in integrated (Grid integration = 3) grid cell

Land use: the model selects the landuse based on the most occurred land use in the area of newly generated grid cell.

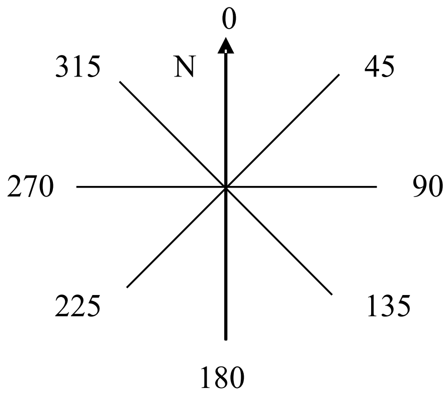


Original land use

New land use in integrated (Grid integration = 3) grid cell

8.1.3 Grid x-axis direction

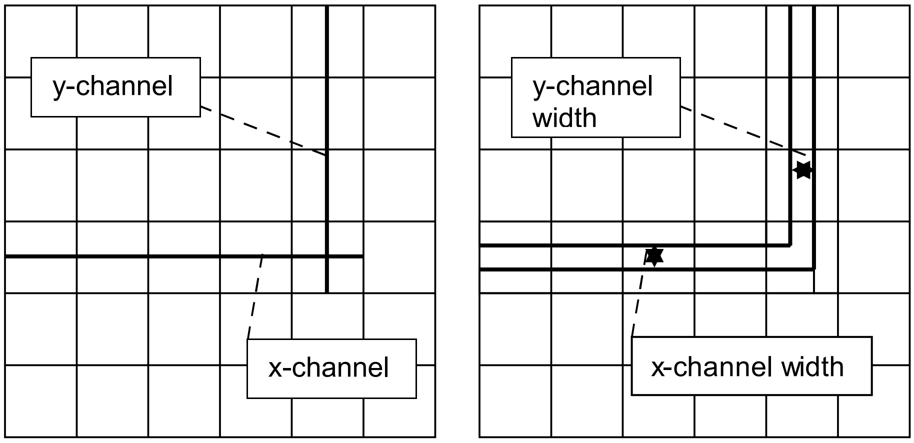
x-axis direction - grid x-axis angle from north to clockwise direction in degrees, typically **90**, as presented below



8.1.4 Channel widths

Single cell width can be changed by user in x- and/or y-direction with the Channel widths tool. Changing the channel width doesn’t change grid structure but only the calculation volume of the selected set of grid cells.

Y-channels are parallel to the y-axis and x-channels correspondingly parallel to the x-axis. The new width defined by using the channel widths tool preserves the grid cell dimensions in its axis direction and changes in other, i.e. in 100 m grid cell the changed x-channel width of 50 m means that the new dimensions of the cell are 100*50 m (x*y). The terms are illustrated below.



Channels can be only either directions of North–South or East–West. The diagonal channels are not possible at this stage. Thus, to channel corners both, x- and y-channels need to be added. It is important to preserve the volume of the channel in the corners and thus, the x- and y-channel widths should be different in corners compared to the straight channel:

- **Straight channel**: if the grid cell size is 100*100m (original horizontal area A_1=10,000m^2) and the user wants to make channel with width of 50 m the new horizontal area for the cell is A_2=50m*100m= 5,000m^2.
- **Corner**: in the corner there are two channels and the horizontal area of those channels should be same than in straight channels. Thus the width of the both channels should be: $w_2=\sqrt{A_2}$ i.e. in this example it should be $w_2=\sqrt{5000m^2}$ = 70.7m.

The user is also able to define different channel depth for each vertical division (see Section 8.1.5 – Vertical division) if the channel's cross section is needed to define in more details.

Changing the channel width impact also the stability of the model and thus, the time steps (see Section 9.6 – Time steps) need to be modified. Thus, it is recommendable not to have much smaller channel widths values compared to the model grid cell size.

Example of the channels in Tonle Sap model application is presented in Figure 14. The depths (or better elevation) of the grid cells which includes channel are also deepened by using the grid depths editor (see Section 8.1.7 –).

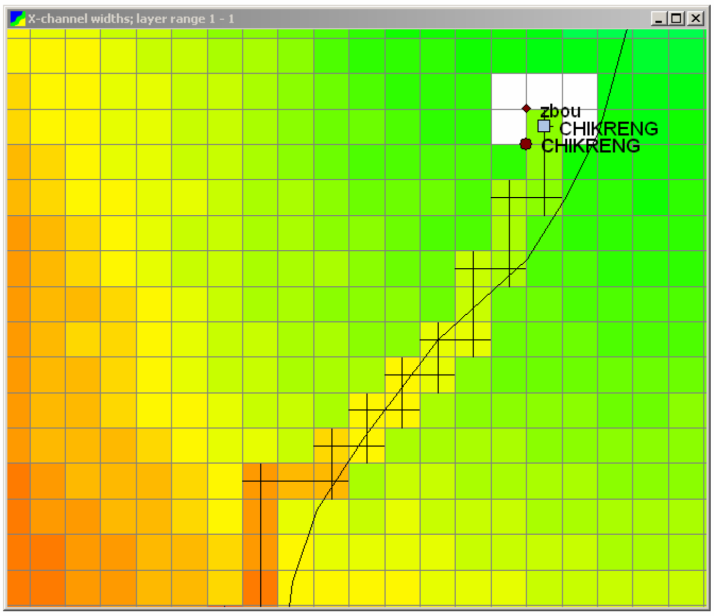
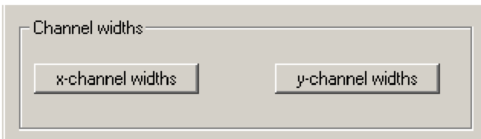


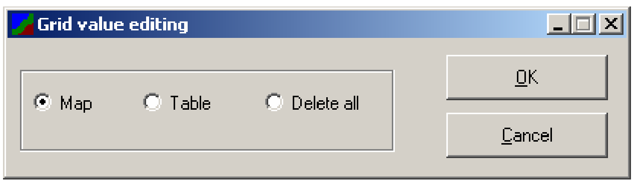
Figure 14. Example of the channel tool used in the Tonle Sap application.

To create or modify channel widths, follow the steps provided below

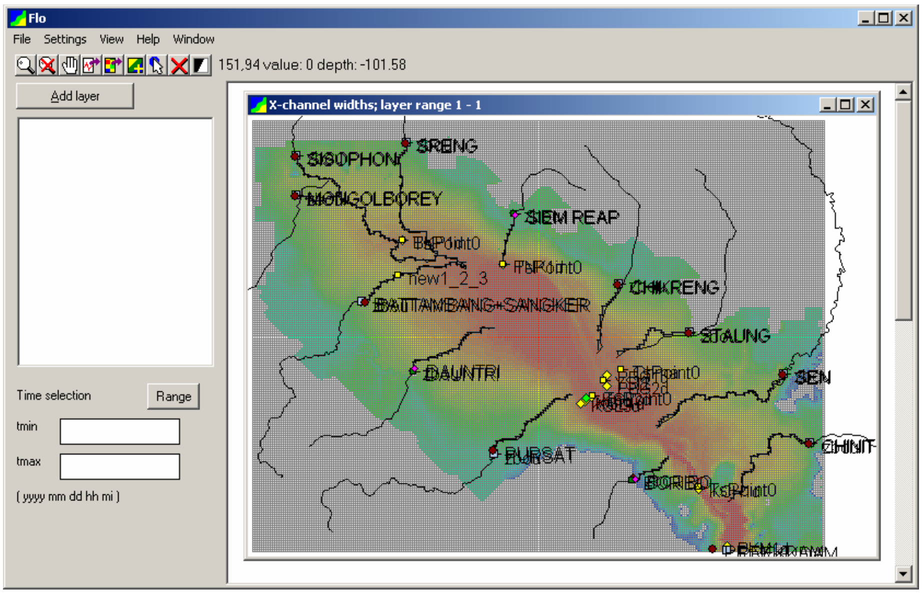
1. Open the Grid parameters window by selecting **Model – Grid parameters…** from the main menu. There is Channel widths sub sector where is button for both, x- and y-channel widths.



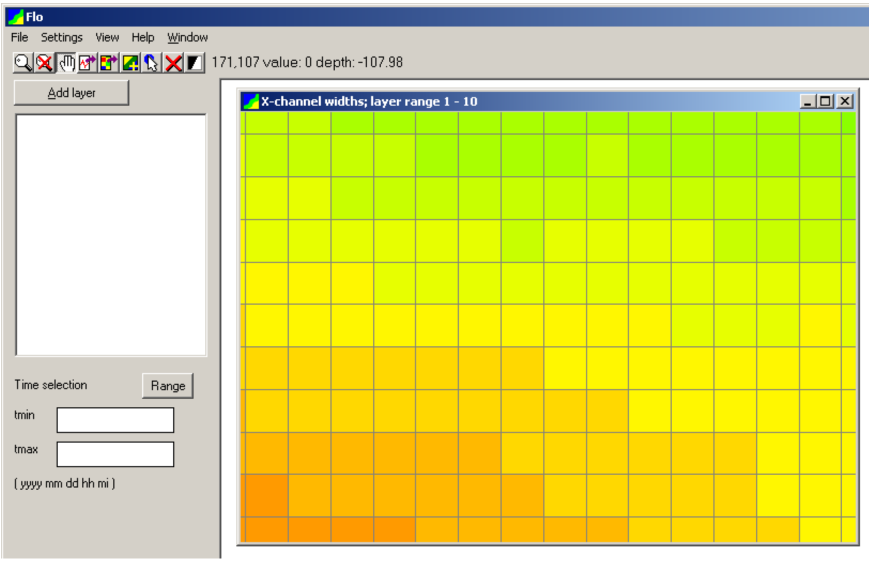
2. There are three options for editing the channel widths value:
    a. Map – edit the channel widths by map interface
    b. Table – edit the channel widths by table based interface
    c. Delete all – delete all the channel width information



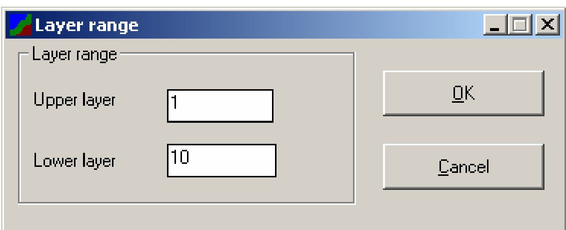
3. **Map:** Select **Map** in the Grid value editing window and press **OK** to edit the channel widths through the map interface.
    a. The following window appears

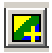


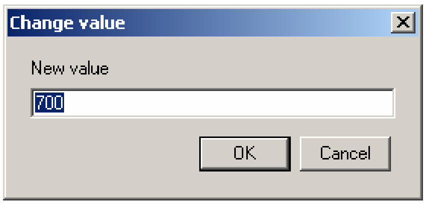
b. First zoom in to the area you want to either edit or add new channel.



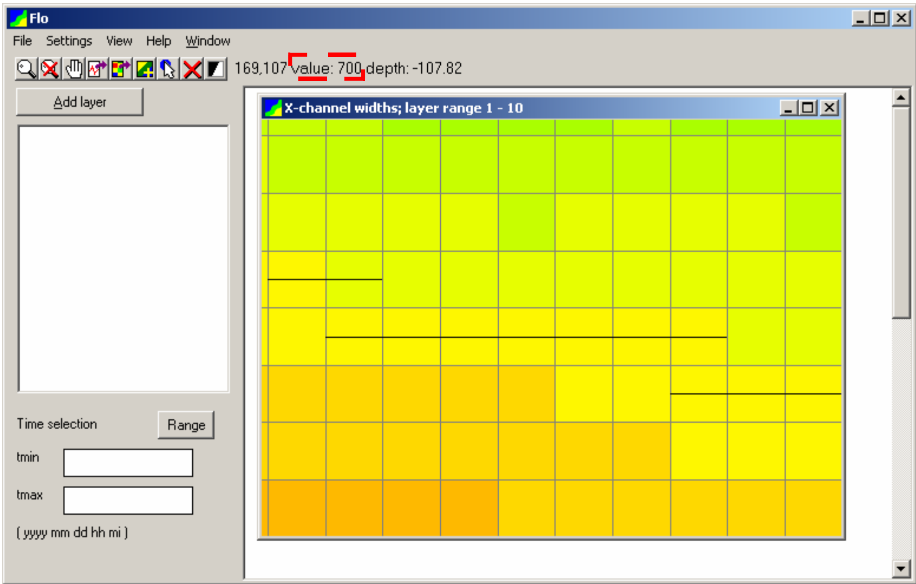
c. The from Settings – Layer range… select the vertical layers you want to change the grid dimensions. If you want to do the changes for all the layers, then check how many vertical divisions there are in the model (see Section 8.1.5 – Vertical division) and use that range: e.g. if you have 10 layers then define the upper and lower layers as illustrated below. If you want different widths for different layers, then define the layers you want to change.



d. Select either continuous  or single mode  from the toolbar to enter the channel widths. With the **Continuous mode** the user can enter the value for the channel width and then draw the channels directly. With the **Single mode** the user needs to define every time separately the value for the channel width.



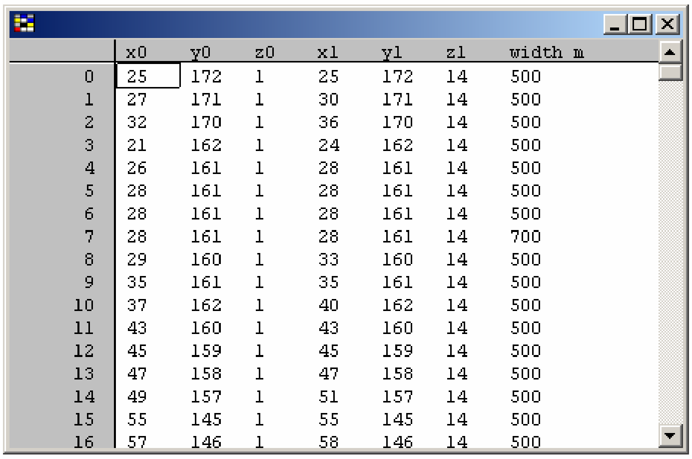
e. Draw the channel to the map and the line describing the place of the channel appears. The value of the channel width can be seen next to the toolbar as illustrated below. Also the grid elevation and coordinate are shown.



f. When you have finished the channel adding, close the channel map window to return to the main model window. This also saves the edits.

4. **Table:** Select **Table** in the Grid value editing window and press **OK** to edit the channel widths through the table based interface.

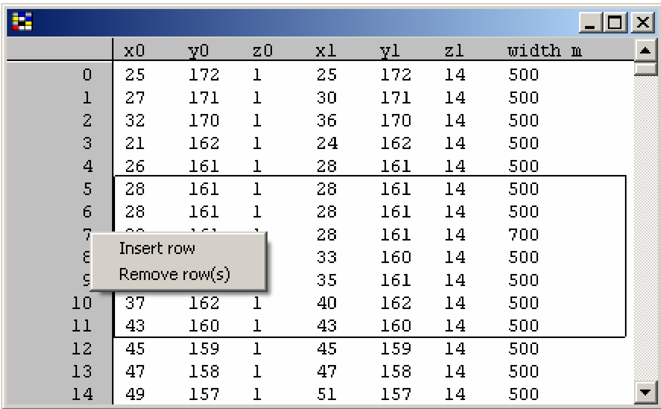
   a. The following table appears



	x0	y0	z0	x1	y1	z1	width m
0	25	172	1	25	172	14	500
1	27	171	1	30	171	14	500
2	32	170	1	36	170	14	500
3	21	162	1	24	162	14	500
4	26	161	1	28	161	14	500
5	28	161	1	28	161	14	500
6	28	161	1	28	161	14	500
7	28	161	1	28	161	14	700
8	29	160	1	33	160	14	500
9	35	161	1	35	161	14	500
10	37	162	1	40	162	14	500
11	43	160	1	43	160	14	500
12	45	159	1	45	159	14	500
13	47	158	1	47	158	14	500
14	49	157	1	51	157	14	500
15	55	145	1	55	145	14	500
16	57	146	1	58	146	14	500

b. First three columns are for the channel's starting point coordinates (x_0, y_0, z_0) and three next for the channel's end point coordinates (x_1, y_1, z_1). Here the channels are defined for vertical layers 1-14. The last column indicates the channel width.

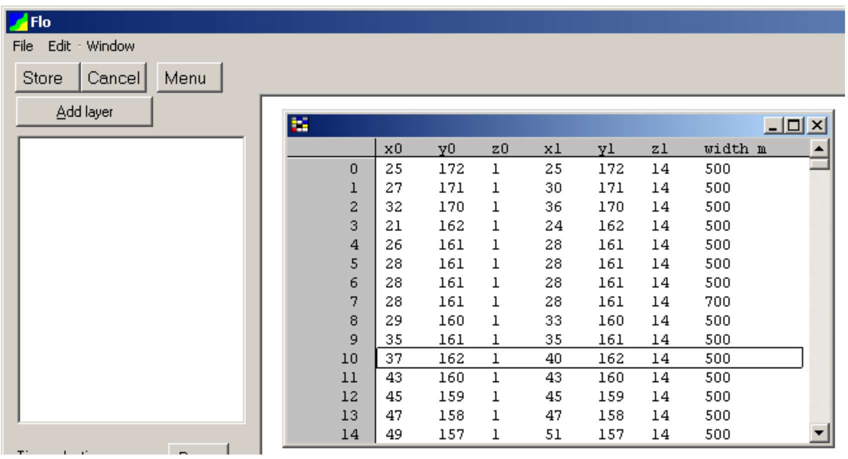
c. User can delete or add layers by selecting first the row to delete or to where to add new by left mouse button. By clicking the right mouse button in the row number you get the menu to insert or remove row(s).



	x0	y0	z0	x1	y1	z1	width m
0	25	172	1	25	172	14	500
1	27	171	1	30	171	14	500
2	32	170	1	36	170	14	500
3	21	162	1	24	162	14	500
4	26	161	1	28	161	14	500
5	28	161	1	28	161	14	500
6	28	161	1	28	161	14	500
7				28	161	14	700
				33	160	14	500
				35	161	14	500
10	37	162	1	40	162	14	500
11	43	160	1	43	160	14	500
12	45	159	1	45	159	14	500
13	47	158	1	47	158	14	500
14	49	157	1	51	157	14	500

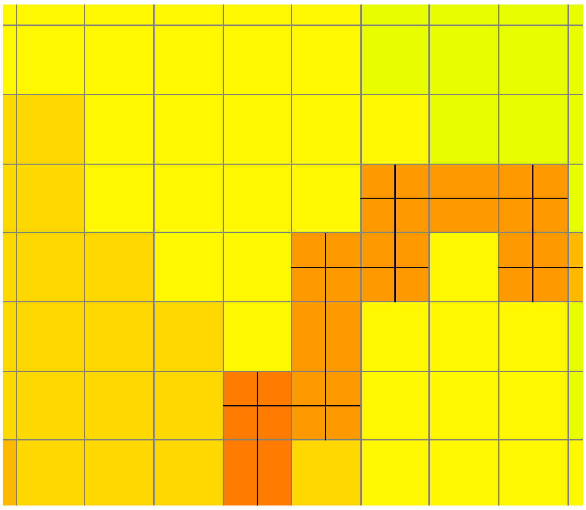
d. User can also edit existing data by clicking the number and typing the new one. Similar technique needs to be used when inserting new data.

e. When you have finished the edits, press **Store** in the toolbar to save the made changes. If you want to cancel the changes, press **Cancel**.



	x0	y0	z0	x1	y1	z1	width m
0	25	172	1	25	172	14	500
1	27	171	1	30	171	14	500
2	32	170	1	36	170	14	500
3	21	162	1	24	162	14	500
4	26	161	1	28	161	14	500
5	28	161	1	28	161	14	500
6	28	161	1	28	161	14	500
7	28	161	1	28	161	14	700
8	29	160	1	33	160	14	500
9	35	161	1	35	161	14	500
10	37	162	1	40	162	14	500
11	43	160	1	43	160	14	500
12	45	159	1	45	159	14	500
13	47	158	1	47	158	14	500
14	49	157	1	51	157	14	500

5. When you have finished the other axis' channel widths, then you have to do the other ones. When you have finished the channel width editing, the channel should look similar to illustrated below.



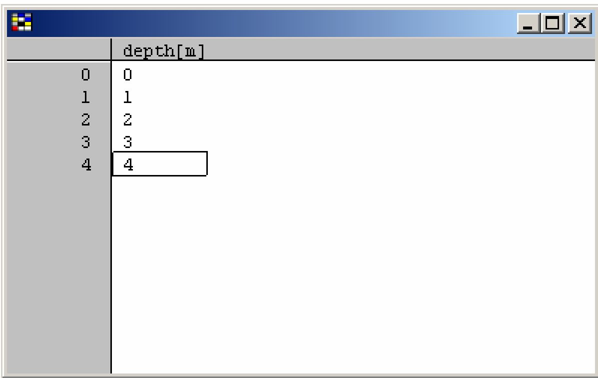
8.1.5 Vertical division

Number of levels - number of grid levels in vertical direction, defined based on the level depths which can be edited as explained below

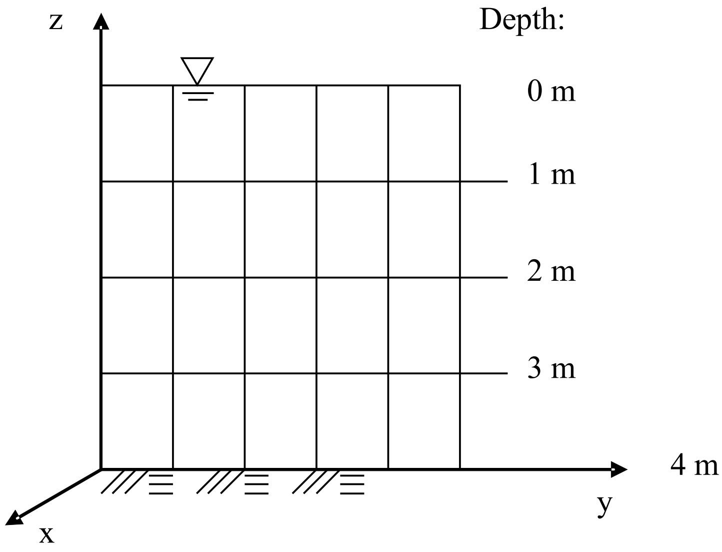
Edit level depths - pops up a matrix in which the level values can be changed

Follow the steps to edit level depths:

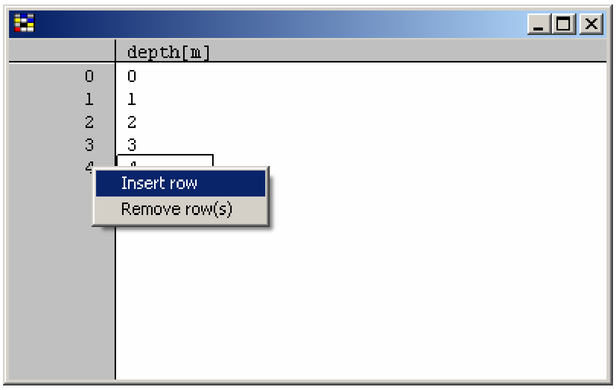
1. Click the "Edit level depths" button and following window appears (sometimes it appears behind the "Grid data" window, you have to drag the window aside)



The 0 value is the water surface and the greatest value is the deepest water level in the model grid as illustrated below



2. You can edit the depths by clicking the depth and typing the new depth. User can also add/delete layers by right the left column and select either **Insert row** to add one layer or **Remove row(s)** to delete row(s)



3. You can also paste the layer depths from e.g. excel file. You can use the **Paste over** function (Menu – Paste over / Ctrl+V) when you have to first you create the number of rows you need. Then copy the depths from excel, select the first depth row and select Paste over. Other option is to use **Paste insert** when the new copied rows are inserted as new rows to the table.
4. Once you have made the required changes to the depth table, press **Store** to save the changes.

User can add basically up to any number of vertical depth layers but normally 100 layers is kept as maximum recommended number of layers. Normal application needs shouldn't be over that either.

Check always the maximum possible water depth in your model grid and enter that to the last depth row. However, the bigger is the maximum depth the smaller is the time step for flow computation. Thus, if you have only few very deep locations you may want to leave these out from computation and select smaller depth to be able to increase the time step, as normally little flow occurs in those exceptionally deep areas.

8.1.6 Depths and volume limits

dmins – minimum grid box depth for velocity grid, usually 0.1-0.3 m

dminz – minimum grid box depth for concentration grid, usually equal to dmins

minvol – minimum grid box volume, usually 0, (raise if computation of transport and diffusion of concentration variables are unstable)

use real volumes – if this is unchecked the contcentration grid cell size will be the basic grid size everywhere; if checked grid size will be adjusted in case of shores and channels; used to enable longer advection timesteps

depth scaling – minimum grid box volume, usually 1

8.1.7 Nesting

Type - nested grid information, values from 1-3
1= nesting type 1; experimental
2= nesting type 2; experimental
3= type 3; use this!

Depth range – defines how nested grid boundaries depths are adjusted; this is used to decrease vertical mixing on the boundary
None = no adjustment
Outer = outer fine grid boundary cells are be adjusted
Outer+zero = as above + land points arel be adjusted
All = all fine grid bounary points will be adjusted to correspond to the coarse grid value
All+zero = same as above + land points are adjusted

Ngrids – number of nested grids (if no nesting, Ngrids=1)

Water divided to whole border – when water is exchanged between the grids, the exchange takes place over the whole dense grid boundary cells; otherwise only outer fine grid cells are involved

Velocity correction on – velocities are corrected on the boundary when water is divided to the whole border; horisontal velocities is each grid cell are corrected on the expense of increased vertical mixing

Edit nest data – you can edit basic nested and non-nested grid data in the following table; manipulation not recommended without good knowledge of the effects

	x0	y0	x1	y1	boxsz	x0	y0	wi	hi	usmth	pycsmth
0	0	0	390	134	200	0	0	78200	27000	0	0

x0, y0 = coordinates of the left corner of the grid (important in nesting)

x1, y1 = coordinates of the right corner of the grid (on non-nested grid defines grid size)

boxsz = size of the grid cell (box size)

x0, y0 = origo coordinates in meters

wi = width of the grid in meters

hi = height of the grid in meters

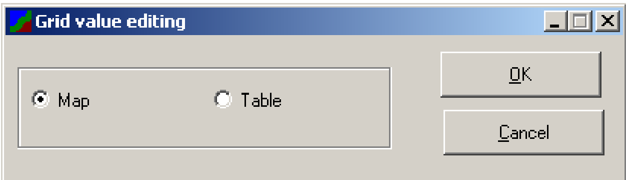
usmth = nested grid dependent surface elevation smoothing number; number tells into how many surrounding cells water amount changes are divided (1 = nearest, 2 = nearest and second nearest etc.); further away surrounding points are less they receive/ contribute water; adds implicity into the model and enables longer external time step; when integer value approaches grid dimensions, dynamic solution approaches implicit solution (see definition of implicity below);

pycsmth = grid dependent smoothing number in pycnocline calculations

> The grid nesting needs to be defined already when creating the model grid. See Section 5.2.3 – Convert *.dig file to 3D grid for more information of the nesting.

8.2 Grid depths

User is able to modify the grid depth data in either map or table based interface. Select **Model – Grid depths**… from main menu to start modifying the grid depths. The following window appears:



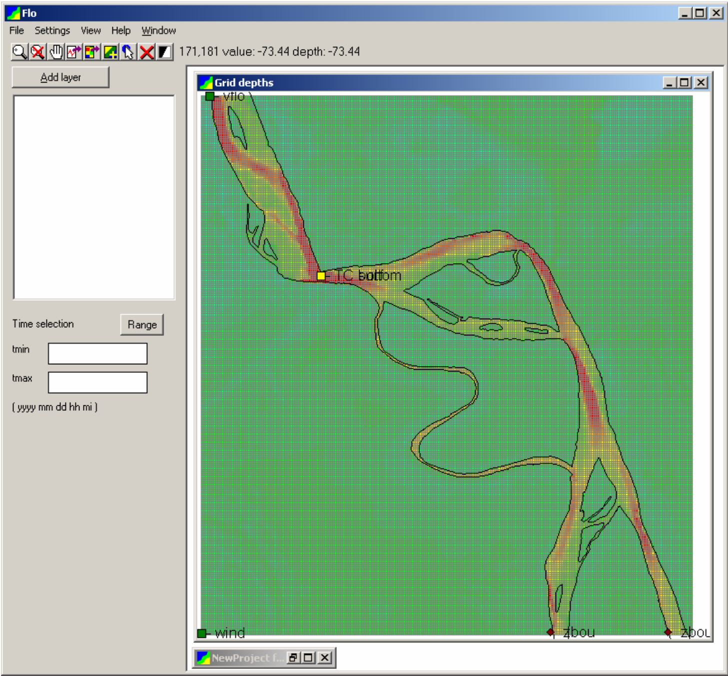
Select either map or table and click **OK**. Go to the following sections to proceed from here:

- Map: Section 8.2.1 – Modifying the grid depths in map
- Table: Section 8.2.2 – Modifying the grid depths in table

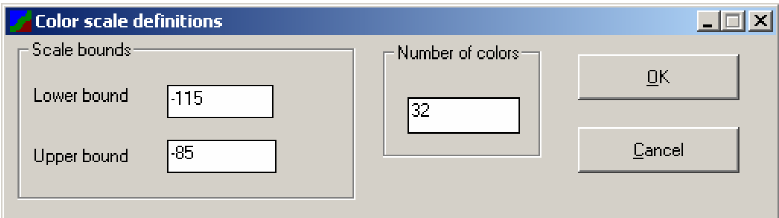
8.2.1 Modifying the grid depths in map

Below the steps to be taken in modifying the grid data in map based interface are declared:

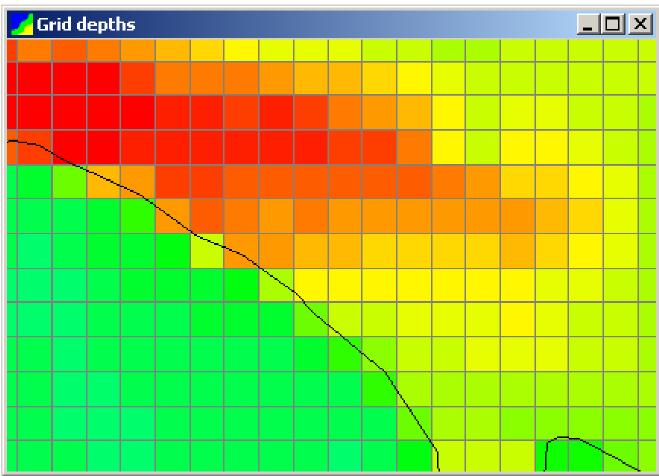
6. Select **Map** in Grid value editing window and click **OK** and the Grid depths window appears.



7. If the colour scale is not right, set up the colour scale by selecting **View – Colour scale…** from the main menu.

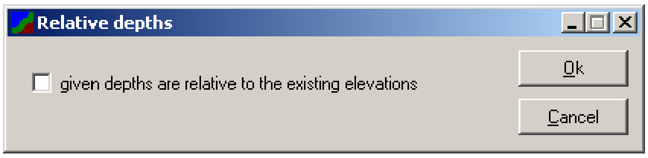


8. The low elevations are shown in red and high ones with the blue colours (this in case that the flood model is used where elevations are negative)
9. Zoom in to the area you want to edit

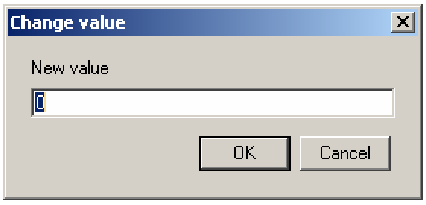


10. Select from the menu **Settings – Relative depths…** to set whether the depths you will be giving are relative to the existing elevations or they are directly the depths related to the datum used in the model. Check on the “given depths are

relative to the existing elevations" and click **OK** if you want to use the relative depths, otherwise click **OK** without checking that option on.



11. Select either continuous  or single mode  from the toolbar to modify the grid depths. With the **Continuous mode** the user can enter the value for the grid depth and then continuously edit the channels. With the **Single mode** the user needs to define every time separately the value for the gird depth.

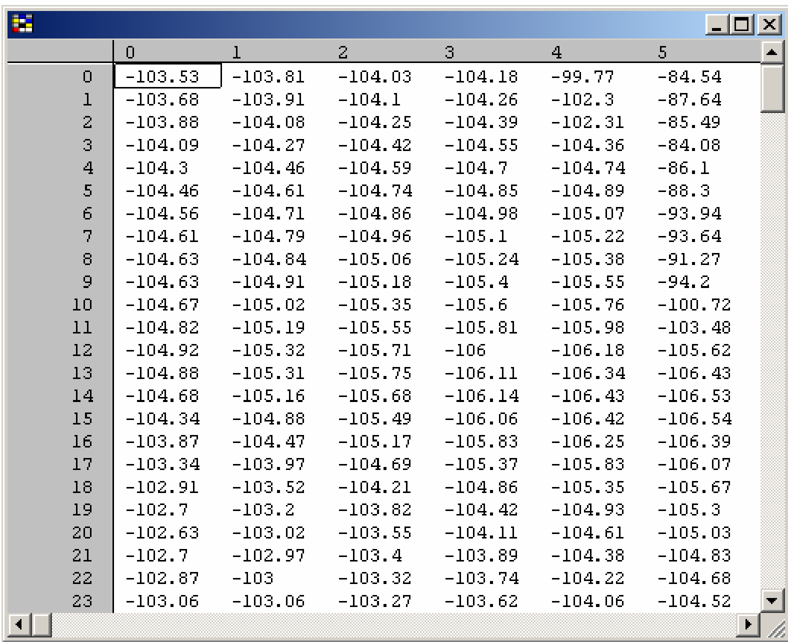


12. now you can modify the grid depths by clicking or dragging the selected grid cells by left mouse button
13. when you have finished the editing, close the Grid depths map window and the modifications will be stored.

8.2.2 Modifying the grid depths in table

Below the steps to be taken in modifying the grid data in table based interface are declared:

1. Select **Table** in Grid value editing window, click **OK** and the Grid depths table appears.



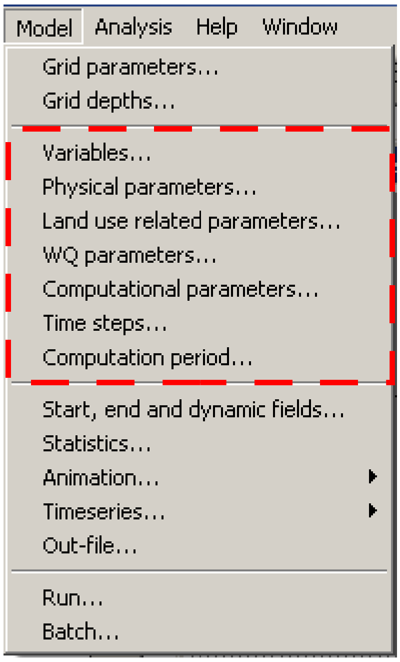
	0	1	2	3	4	5
0	-103.53	-103.81	-104.03	-104.18	-99.77	-84.54
1	-103.68	-103.91	-104.1	-104.26	-102.3	-87.64
2	-103.88	-104.08	-104.25	-104.39	-102.31	-85.49
3	-104.09	-104.27	-104.42	-104.55	-104.36	-84.08
4	-104.3	-104.46	-104.59	-104.7	-104.74	-86.1
5	-104.46	-104.61	-104.74	-104.85	-104.89	-88.3
6	-104.56	-104.71	-104.86	-104.98	-105.07	-93.94
7	-104.61	-104.79	-104.96	-105.1	-105.22	-93.64
8	-104.63	-104.84	-105.06	-105.24	-105.38	-91.27
9	-104.63	-104.91	-105.18	-105.4	-105.55	-94.2
10	-104.67	-105.02	-105.35	-105.6	-105.76	-100.72
11	-104.82	-105.19	-105.55	-105.81	-105.98	-103.48
12	-104.92	-105.32	-105.71	-106	-106.18	-105.62
13	-104.88	-105.31	-105.75	-106.11	-106.34	-106.43
14	-104.68	-105.16	-105.68	-106.14	-106.43	-106.53
15	-104.34	-104.88	-105.49	-106.06	-106.42	-106.54
16	-103.87	-104.47	-105.17	-105.83	-106.25	-106.39
17	-103.34	-103.97	-104.69	-105.37	-105.83	-106.07
18	-102.91	-103.52	-104.21	-104.86	-105.35	-105.67
19	-102.7	-103.2	-103.82	-104.42	-104.93	-105.3
20	-102.63	-103.02	-103.55	-104.11	-104.61	-105.03
21	-102.7	-102.97	-103.4	-103.89	-104.38	-104.83
22	-102.87	-103	-103.32	-103.74	-104.22	-104.68
23	-103.06	-103.06	-103.27	-103.62	-104.06	-104.52

2. Every grid cell has one value in the matrix. User can locate the grid in the map by using the grid coordinates to calculate the place in matrix. The (0, 0) value in the depth matrix is the upper right corner in the grid.
3. User can edit existing data by clicking the number and typing the new one. Similar technique needs to be used when inserting new data.
4. When you have finished the edits, press **Store** in the toolbar to save the made changes. If you want to cancel the changes, press **Cancel**.

8.2.3 Adding the channel information

See Section 8.1.4 – Channel widths for detail information.

9 MODEL VARIABLES



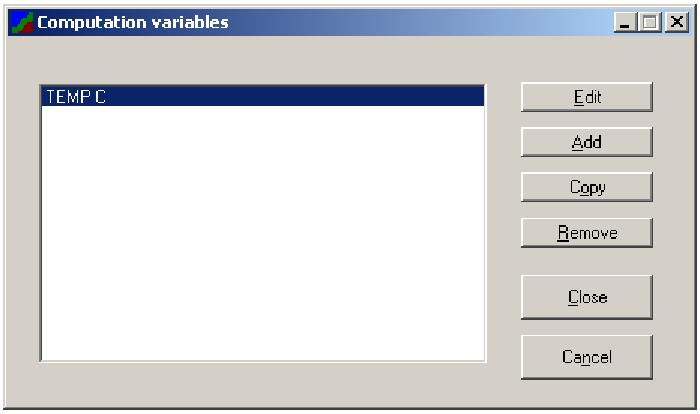
This chapter deals with the model variables. It goes through each menu item and presents the variables of those.

The chapter is divided to seven parts:

9.1 Variables
9.2 Physical parameters
9.3 Landuse parameters
9.4 Water quality parameters
9.5 Computational parameters
9.6 Time steps
9.7 Computation period

9.1 Variables

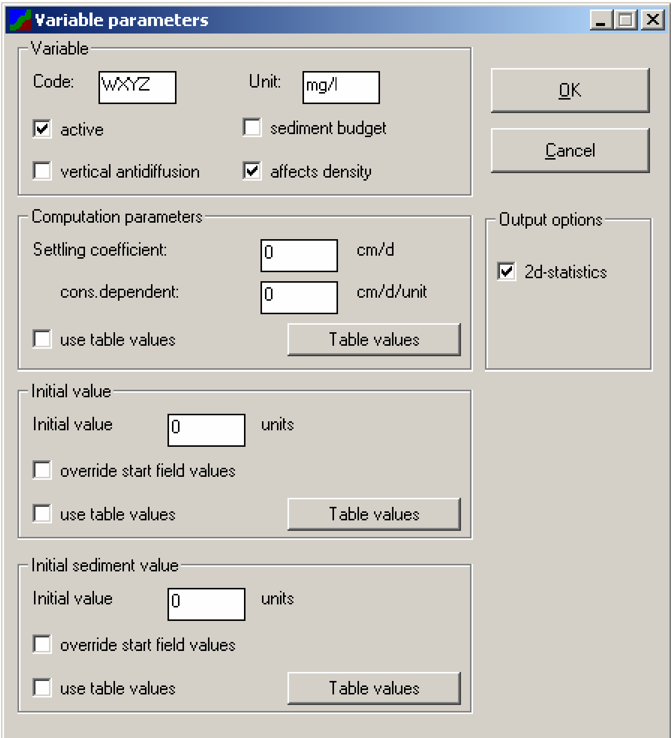
In variable window you can add the concentration variables (e.g. temperature, suspended sediment, oxygen, etc). Select **Model – Variables…** to open the Computation variables window as shown below



Following actions can be selected from the list in right part of the window:

- Edit - edit item data
- Add - add new item
- Copy - create a new item by copying an existing item
- Remove - remove an item
- Close - close the list window

When user is either editing or creating new variable the following "Variable parameters" window appears (this example window appears when creating new variable):

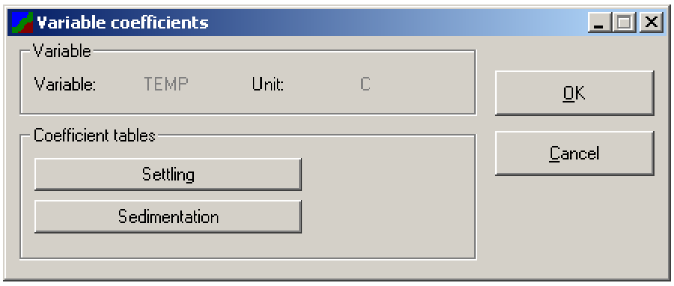


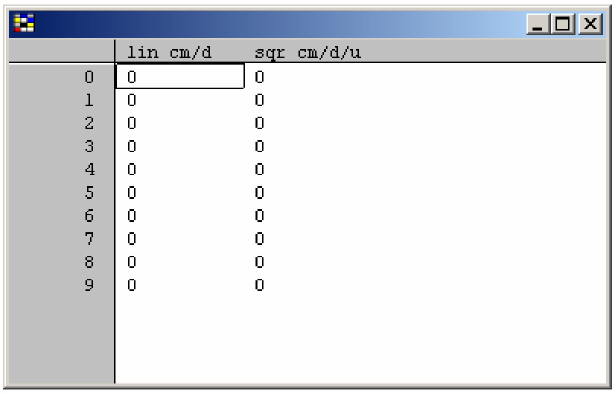
In Variable section of the window the following parameters can be defined:

- Code - variable name code, 4 uppercase letters, below some of the most common name codes with their Unit
    - TEMP = temperature [C]
    - SALI = salinity [mg/l]
    - SEDI = suspended sediment [mg/l]
    - OXYG = dissolved oxygen [mg/]
- Unit – unit for the variable, either mg/l for concentration or C for temperature
- Active - checked if the variable is computed, unchecked if not
- Vertical antidiffusion - computational method for reducing numerical diffusion in vertical direction, not normally used
- Sediment budget – checked if sedimentation of the variable is needed to be calculated (normally on for suspended sediment)
- Affects density – checked if the variable is wanted to affect density, as salinity

The following computational parameters can be defined

- Settling coefficient - linear settling coefficient, cm/d
- Cons.dependent - nonlinear concentration dependent settling coefficient, unit/cm/d
- Use table values - settling coefficients can be separately given for each grid depth layer, is this checked the individual layer values are used
- Table values - pops up a matrix containing separate settling coefficients for each vertical grid layer





In Initial value section user is able to define the initial values for the variable.

- Initial value - one initial value for all the grid area
- Override start field values – model will use values defined here instead of the ones from the start field (see Section 10.1 – Start, end and dynamic fields)
- Use table values - several initial values for different rectangular areas can be given. Check this if you want to define more than one initial value area. Define the values for each area in Table values.
- Table values - pops up a window where two options to give the initial values can be used
    - Table: matrix into which initial value areas can be entered. The matrix contains a one line for each rectangular initial value area. On the line are x0,y0,z0,x1,y1,z1 and c0 values, where the coordinates define the area and c0 is the preferred initial value.
    - Map: user can define the values based on the location in the map

In Initial sediment value section user is able to define the initial values for the amount of variable that exist in the bottom of the water body when the calculation starts.

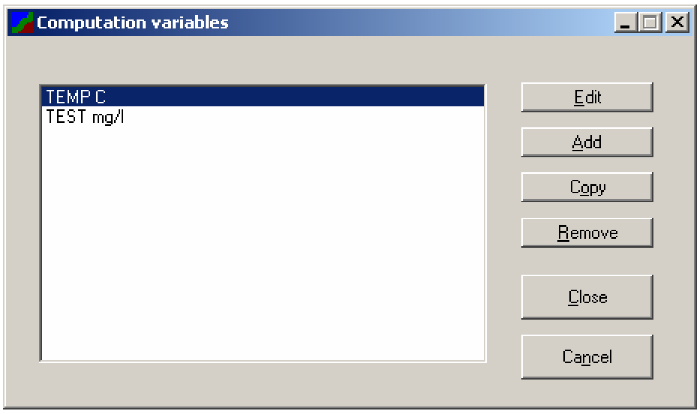
- Initial value - one initial value for all the grid area
- Override start field values – model will use values defined here instead of the ones from the start field (see Section 10.1 – Start, end and dynamic fields)
- Use table values - several initial values for different rectangular areas can be given. Check this if you want to define more than one initial value area. Define the values for each area in Table values.
- Table values - pops up a window where two options to give the initial values can be used
    - Table: matrix into which initial value areas can be entered. The matrix contains a one line for each rectangular initial value area. On the line

are x0,y0,z0,x1,y1,z1 and c0 values, where the coordinates define the area and c0 is the preferred initial value.

- Map: user can define the values based on the location in the map

The 2d-statistics should be checked in the Output options part of the window if user wants to include the parameter into the 2d-field output.

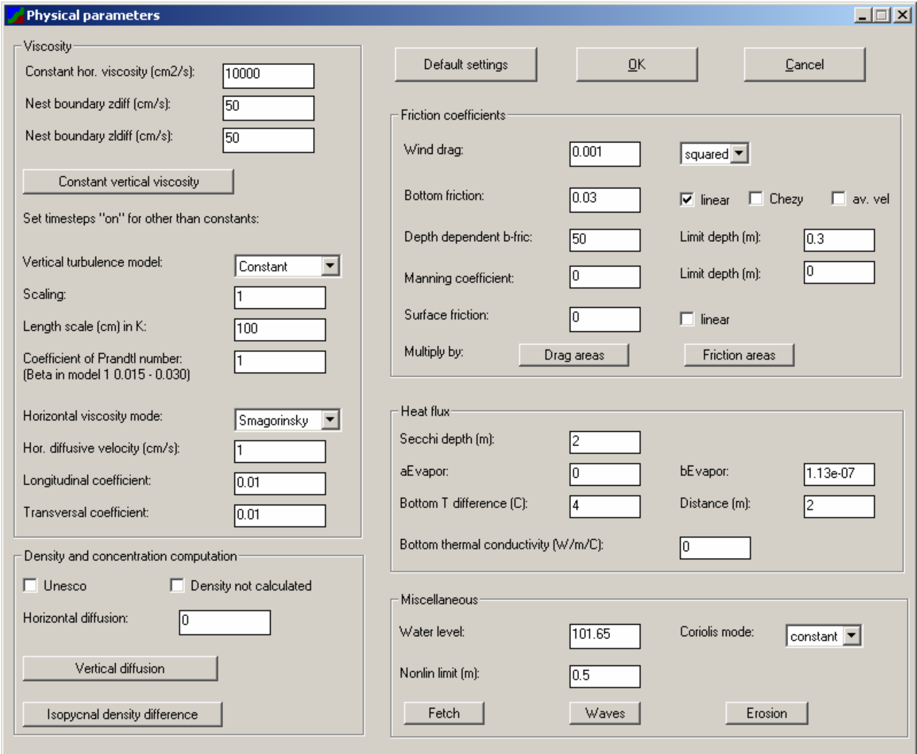
Press OK to save the settings and to get back to the “Computation variables” window. The new variable, if added one, can be seen in the list as shown below (TEST variable):



9.2 Physical parameters

User can define the different physical parameters for computation in **Physical parameters** window. The window is divided to five sub-categories:

- Viscosity
- Density and concentration computation
- Friction coefficients
- Heat flux
- Miscellaneous



The parameters and options for each sub-category are defined in more details in the following sections.

9.2.1 Viscosity

Theory and model realisation of turbulence and viscosity and presented in WUP-FIN Model Report and in Manual Chapter 26. Here only a short description of parameters is given.

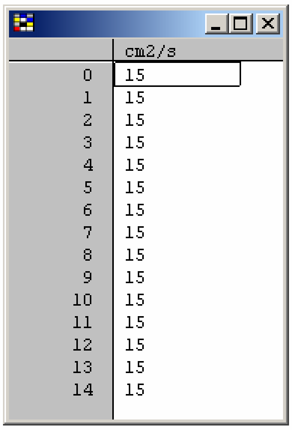
Viscosity is a measure of the resistance of a fluid to deformation under shear stress. It is commonly perceived as "thickness", or resistance to pouring. Viscosity describes a fluid's internal resistance to flow and may be thought of as a measure of fluid friction. Thus, water is "thin", having a low viscosity, while vegetable oil is "thick" having a high viscosity.

[Source: Wikipedia (www.wikipedia.org)]

Basic variables:

- Constant horizontal viscosity [cm^2/s]: when horisontal viscosity is not calculated one needs to specify it; appropriate values range from 1000 – 100000 depending on grid size
- Nest boundary zdiff [cm/s]: added surface diffusion on nested grid boundaries; required in some cases to stabilise flow; should be as small as possible, value 50 is often used
- Nest boundary zldiff [cm/s]: corresponding nested boundary isopycnal diffusivity

Constant vertical viscosity: when vertical viscosity is not computed prescribe viscosity over layer boundaries; typical values range between 15 (relatively calm lakes and sea areas) to 100 (high velocity river flow) cm^2/s.



	cm2/s
0	15
1	15
2	15
3	15
4	15
5	15
6	15
7	15
8	15
9	15
10	15
11	15
12	15
13	15
14	15

Kinematic viscosity (Greek symbol: ν) has SI units ($m^2 \cdot s^{-1}$). The cgs physical unit for kinematic viscosity is the stokes (abbreviated S or St), named after George Gabriel Stokes. It is sometimes expressed in terms of centistokes (cS or cSt). In U.S. usage, stoke is sometimes used as the singular form. In environmental fluid flow kinematic viscosity is insignificant compared to turbulence.

1 stokes = 100 centistokes = 1 $cm^2 \cdot s^{-1}$ = 0.0001 $m^2 \cdot s^{-1}$.

[Source: Wikipedia (www.wikipedia.org)]

Vertical turbulence models:

- Constant
- Wind, cor., depth: wind, Coriolis and depth dependent model; only for experimentation
- Richardson: Richardson model; only for experimentation
- L, Rich., vdiff: Richardson + length scale; only for experimentation
- k-e: k-epsilon; calculation of turbulent kinetic energy and its dissipation; most universal and accurate
- K+L: kinetic energy and length scale calculation; appoximates k-e in a stationary case
- K: kinetic energy, length scale given (see below)

Parameters:

- Scaling: viscosity sclaing coefficient in case viscosity is calculated
- Length scale [cm] in K:
- Coefficient of Prandtl number

Turbulence: In fluid dynamics, turbulence or turbulent flow is a flow regime characterized by chaotic, stochastic property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time. Flow that is not turbulent is called laminar flow. The (dimensionless) Reynolds number characterizes whether flow conditions lead to laminar or turbulent flow.

[Source: Wikipedia (www.wikipedia.org)]

Horizontal viscosity mode:

In contrast to homogenous classical horisontal viscosity formulation presented above

- Smagorinsky
- lin. x-y: viscosity in x- and y-directions
- all dir: experimental in flow direction
- x-y and diag.: experimental in x-, y- and diagonal directions
- velocity diff.: experimental velocity difference dependent.

Horizontal viscosity parameters:

- Hor. diffusive velocity [cm/s]: viscosity coefficient in other modes than Smagorinsky
- Longitudinal coefficient: viscosity scaling in x-direction (originally in flow longitudinal direction); in the Smagorinsky formula this parameter is $c_H \frac{\sqrt{2}}{4}$ in the x-direction (see below); use 0.1 or less for Smagorinsky; in litterature indication is given that c_h values 0.1 – 0.2 work best corresponding to model parameter values 0.04 – 0.07
- Transversal coefficient: viscosity scaling in y-direction (originally in flow transversal direction); in the Smagorinsky formula this parameter is $c_H \frac{\sqrt{2}}{4}$ in the y-direction (see below); use 0.1 or less for Smagorinsky; values in range 0.04 – 0.07 may work best.

Smagorinsky formula is:

$$D_h = c_H \frac{\sqrt{2}}{4} \Delta x \Delta y \sqrt{2\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2}$$

c_h is dimensioneless HORCON parameter. Advantages of the Smagorinsky formulation are that c_h is non-dimensional, viscosity decreases as grid cell size decreases and viscosity is small if velocity gradients are small.

9.2.2 Density and concentration computation

Options:

- Unesco: use standard Unesco formula for density; if this is not checked a simplified square law for temperature dependent density is used and salinity or any dissolved substance dependt density is added to this
- Density not calculated

Variables:

- Horizontal diffusion: horizontal diffusion coefficient for concentration calculation (see chapter 24.3)
- Vertical diffusion: vertical concentration diffusion coefficient; prescribed over each model layer boundary
- Isopycnal density difference: initial density difference over each layer boundary in isopycnal density calculation mode

Diffusion, being the spontaneous spreading of matter (particles), heat, or momentum, is one type of transport phenomena. Diffusion is the movement of particles from higher chemical potential to lower chemical potential (chemical potential can in most cases of diffusion be represented by a change in concentration).

Momentum diffusion refers to the diffusion, or spread of momentum between particles of matter, usually in the liquid state. In the case of the laminar flow of a liquid past a solid surface, momentum diffuses across the boundary layer near the surface. The gradient in this case occurs between the liquid in contact with the surface, which does not move at all and has zero momentum, and the liquid far away from the wall, which has momentum proportional to the speed at which it is flowing. The rate of transport is governed by the viscosity of the fluid and the momentum gradient.

[Source: Wikipedia (www.wikipedia.org)]

Isopycnal is a surface of constant water density. In the ocean, as the depth increases, so too does the density. Varying degrees of salinity and temperature act to modify the density of water, and the denser water always lies below the less dense water. Because of the action of winds and ocean currents, isopycnals are not always level.

[Source: Wikipedia (www.wikipedia.org)]

In the model isopycnal solution mode describes density differences as dynamic isosurfaces eliminating this way numerical diffusion.

9.2.3 Friction coefficients

Friction is the force that opposes the relative motion or tendency of such motion of two surfaces in contact. It is not, however, a fundamental force, as it originates from the electromagnetic forces between atoms. Friction between solid objects and gases or liquids is called fluid friction.

The coefficient of friction (also known as the frictional coefficient) is a dimensionless scalar value which describes the ratio of the force of friction between two bodies and the force pressing them together.

[Source: Wikipedia (www.wikipedia.org)]

Wind drag:

The wind drag coefficient is not only a physical parameter, but adjusting coefficient enables wind measured in some other place than just over water surface to be used in the model. The effective wind drag (or friction) coefficient value depends on where and on what height wind is measured. Because near ground wind velocity increases sharply in upward direction, the drag coefficient has to be set lower the higher the measurement. In similar way wind speed is in general higher over lake or sea area than over forested area and wind drag needs to be increased when wind is measured over forested or land area in general. Typical values are 0.001 - 0.002. Use allways square (physical shear stress) form of the wind drag, other options are for testing purposes only. See also chapter 24.1 Wind Shear Stress in Part II of the Manual.

Wind drag depends on water surface roughness/ waves. Although model calculates sheltering, fetch and waves, these don't have at the moment connection to the wind drag.

Bottom friction:

Optional bottom friction coefficients and equations:

- Linear: linear form of bottom friction
- Chezy: Chezy law used for bottom friction utilising bottom roughness
- Av. Vel: friction velocity is obtained from

If non of these is selected square law is used. The form of the bottom friction term in momentum equations is $-c_b \cdot u \cdot \sqrt{u^2+v^2}, -c_b \cdot v \cdot \sqrt{u^2+v^2}$ for bottom flow x- and y-components. c_b is the bottom friction coefficient. Quadratic formula is physically justified shear stress form and works best in most of cases, at least for 3D flow modelling. Linear form can be used in some cases to stabilise simulation in case of extremely high friction such as flow through vegetation (for instance on Tonle Sap floodplain). Typical non-linear bottom friction coefficient value range is 0.0025 – 0.01 depending on the bottom roughness and flow characteristics. Linear bottom friction can be set around 0.03.

In 3D applications bottom friction and vertical viscosity are connected: by increasing vertical viscosity average flow “feels” bottom friction more effectively. This impacts also water elevations. One can interpret vertically averaged flow calculation as 3D calculation with infinite viscosity. Because of this bottom friction value calibrated with 2D model option is not in general appropriate for 3D calculation. For further discussion of bottom friction reader is reffered to chapter 24.2.

Depth dependent b-fric:

With this option bottom friction coefficient increases as a function of decreasing depth: $c_b(z)=c_0/z$. User must specify limit depth until which the coefficient increases. Observe that the depth dependent friction is *added* to the basic bottom friction.

Depth dependent bottom friction can stabilise flow in case of thin water layer is flooding and drying floodplain. In such cases it is possible that in some grid point water level starts to oscillate between dry and wet states. Increased friction can stabilise flow in such cases. Also depth dependent bottom friction can describe effect of increased viscosity and average friction (see discussion in “Bottom friction” above) with decreasing depth.

Manning coefficient:

This parameter value determines wheter diffusion wave approximation (for the flux in x-direction $q_x=\frac{z^{5/3}}{n}\frac{\partial z}{\partial x}$, where n is the Manning coefficient and z water depth). When Manning coefficient is different from zero diffusion wave approximation is used for calculation when water depth is less or equal than a given limit depth. The rationale behind the use of diffusion wave approximation is, that with thin sheet flow over floodplain with high roughness Navier-Stokes equations don't describe the flow accurately and diffusion wave approximation is more relevant then. The latest Mekong applications don't utilise diffusion wave solution method.

The Manning formula is an empirical formula for open channel flow, or flow driven by gravity. It was developed by the Irish engineer Robert Manning.

$$V=\frac{1}{n}R_h^{\frac{2}{3}}\cdot S^{\frac{1}{2}}$$

Where:

V - cross-sectional average velocity (m/s)

n - Manning coefficient of roughness

Rh - hydraulic radius (m)

S - slope of pipe/channel (m/m)

[Source: Wikipedia (www.wikipedia.org)]

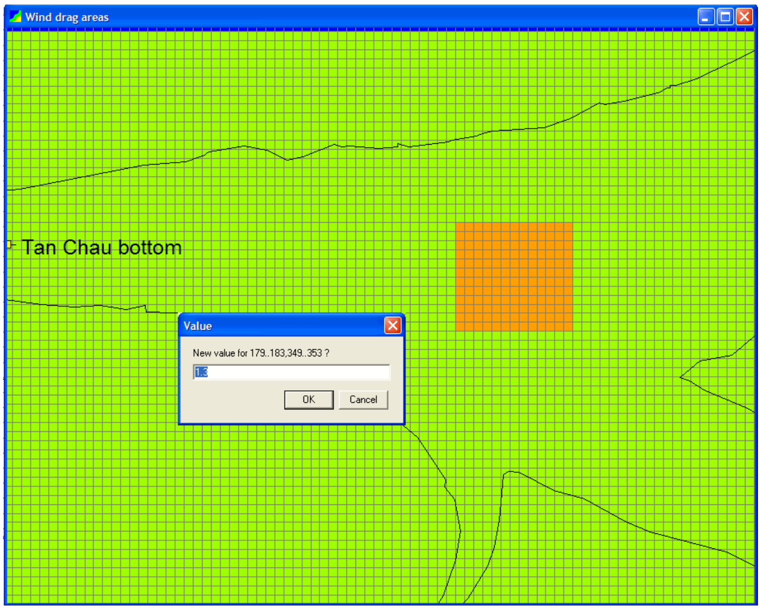
Typical verified manning coefficient value range for rivers is 0.02 – 0.08 depending on the river characteristics.

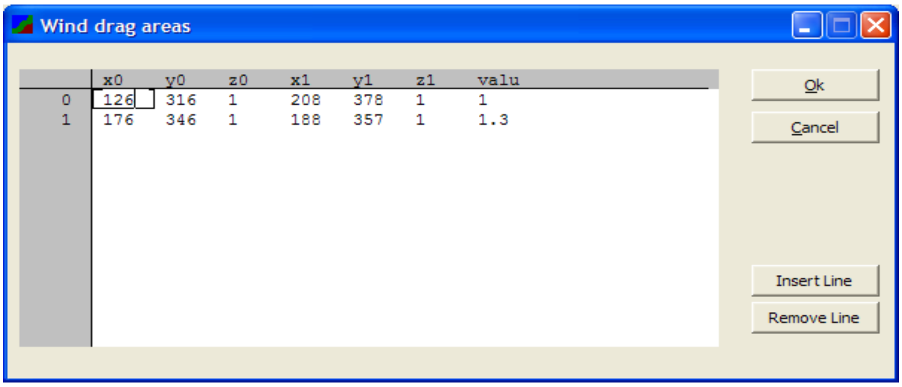
Surface friction

Surface friction is used for specifying ice friction. It is analogous for bottom friction.

Multiply by:

It is possible to change wind drag or bottom friction coefficients in any area by multiplying them with a scaling coefficient. The scaling coefficients can be prescribed by editor or by spreadsheet:



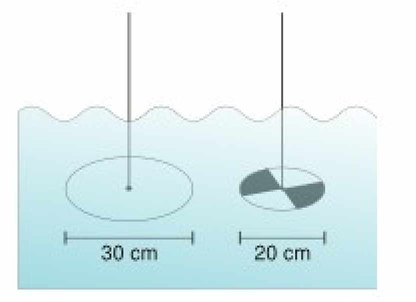


9.2.4 Heat flux

Heat fluxes are calculated when modelling water temparature.

Secchi depth: See box below for definition of Secchi depth. It is used for determining light penetration into the water column.

Secchi depth: Secchi disk is a device used to measure water transparency in open waters of lakes, bays, and the ocean. The pattern shown in the image is drawn or painted onto a card or acrylic, mounted on a pole or line, and lowered slowly down in the water. The depth at which the pattern on the disk is no longer visible is taken as a measure of the transparency of the water. This measure is known as the *Secchi depth* and is related to water turbidity.



[Source: Wikipedia (www.wikipedia.org)]

Evaporative heat transfer can be significant. In the model there are two parameters connected to it:

aEvapor [m s^{-1} mb^{-1}]: free convention factor in the evaporative heat transfer; depends on virtual temperature gradient; often set to 0

bEvapor: coefficient for the wind dependent forced convection factor; recommended value 1.13 10^{-9} mb^{-1}; depends eg. on fetch.

Bottom T difference [C]: temperature gradient between water column and bottom for bottom heat transfer

Distance [m]: Distance over which temperature gradient is measured or estimated

Bottom thermal conductivity (W/m/C): see box below.

Bottom thermal conductivity: In physics, thermal conductivity, k, is the intensive property of a material that indicates its ability to conduct heat.

It is defined as the quantity of heat, Q, transmitted through a thickness L, in a direction normal to a surface of area A, due to a temperature difference ΔT, under steady state conditions and when the heat transfer is dependent only on the temperature gradient.

thermal conductivity = heat flow rate × distance / (area × temperature difference)

$k = Q \times L / (A \times \Delta T)$

[Source: Wikipedia (www.wikipedia.org)]

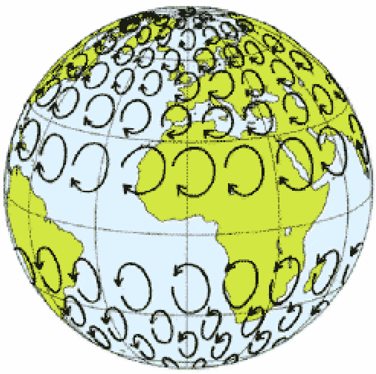
9.2.5 Miscellaneous

Water level [m]: Initial water level. The values depend on the model vertical reference system.

Coriolis mode:

- none: no Coriolis (for testing purposes only)
- constant: constant over the calculation domain
- changing: changes according to latitude; necessary for large areas

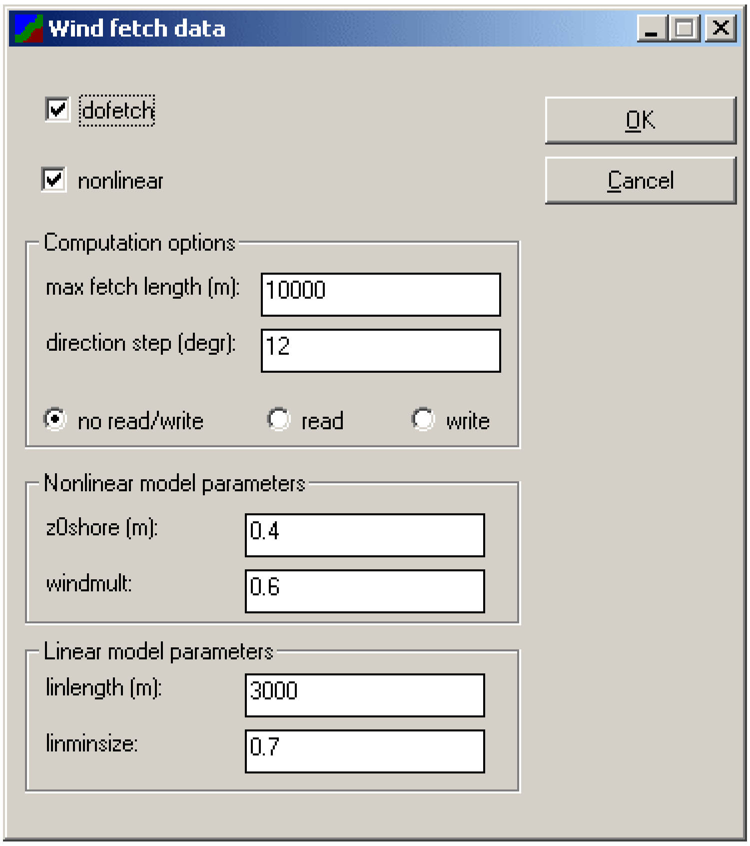
Coriolis: The Coriolis effect is an apparent deflection of a moving object in a rotating frame of reference. There are examples of this effect in everyday life, such as the direction of rotation of cyclones. Due to the effect, cyclones rotate counterclockwise in the northern hemisphere, and clockwise in the southern hemisphere.



[Source: Wikipedia (www.wikipedia.org)]

Nonlin limit [m]: depth limit for calculation of momentum advection; sometimes required to stabilise thin sheet flows over floodplain

Fetch:



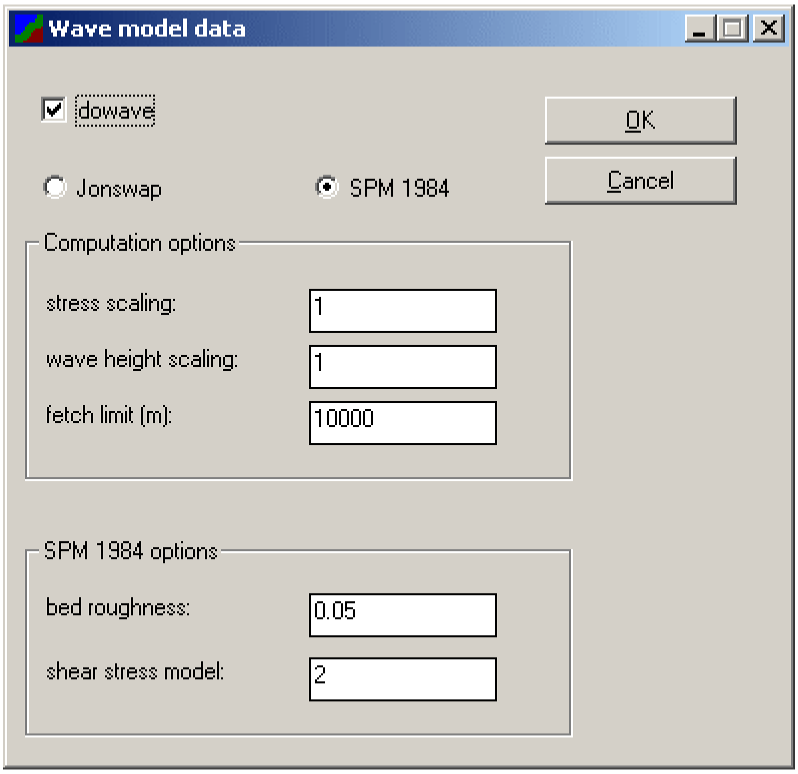
- dofetch: if selected fetch calculation is on
- nonlinear: if selected nonlinear form is selected (default)
- **Compuation options**
- max fetch lentgh [m]: limit until which fetch increases
- direction step [degr]: what is the difference in degrees between different wind directions for which fetch is calculated; in-between calculated wind directions fetch is interpolated
- no read/write, read, write: determines wheter fetch is read, written on none; storing fetches and using them consequently saves calculation time, but requries quite lot of disk space
- **Nonlinear model parameters**
- z0shore [m]: shore roughness
- windmult: wind stress scaling
- **Linear model parameters**
- linlength [m]: length where wind is assumed to reach its prescribed value after sheltering effect of the shore
- linminsize: start ratio to fully developed stress

Fetch, often called the fetch length, is a term for the length of water over which a given wind has blown. It is used in geography and meteorology and is usually associated with coastal erosion. It plays a large part in longshore drift as well.

The fetch length along with the wind speed determines the size of waves produced. For example, the winds which travel from the East Coast of the United States and hit the west coast of Ireland would have an extremely large fetch and would produce very large waves if the wind speed was also high.

[Source: Modified from Wikipedia (www.wikipedia.org)]

Waves:



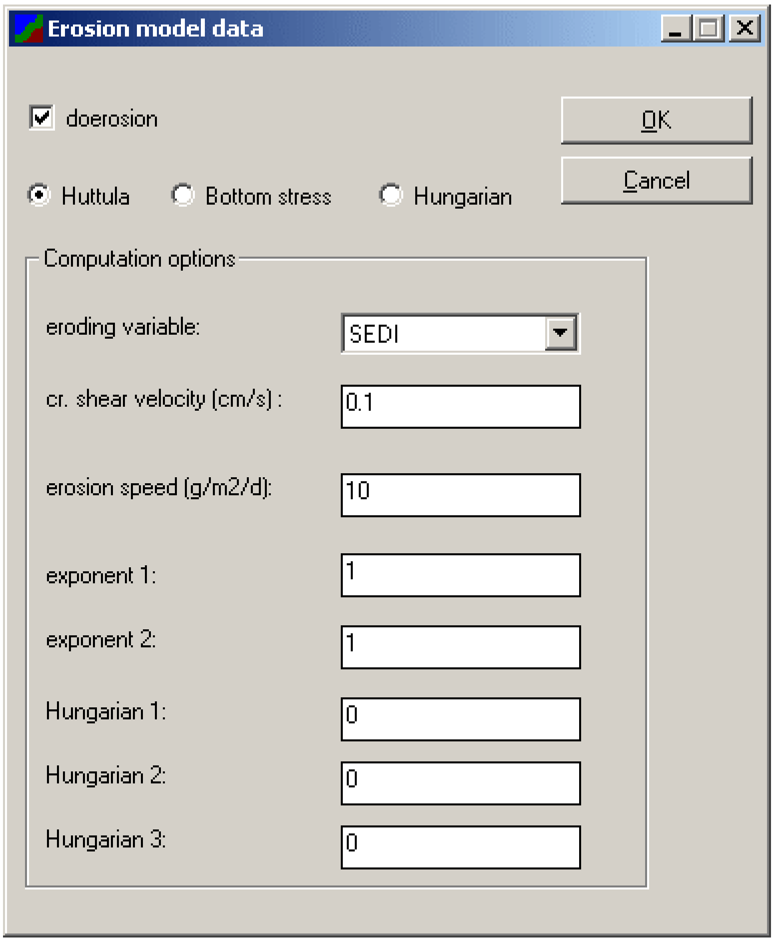
Tonle Sap waves are the main bed mobilization force. Because the main wave impact happens during shallow water periods, a wave model must be capable of handling shallow water conditions. The shallow water conditions pose a specific problem for wave modeling because waves "feel" bottom. The SPM1984 model has been developed specifically for shallow water conditions. Jonswap model is suitable for deep water waves only. The wave model parameters are:

- **Computation options**
- stress scaling: coefficient for calculated wind stress
- wave height scaling: coefficient for calculated wave heights
- fetch limit [m]: fetch limit after which wave energy doesn't increase
- **SPM 1984 options**
- bed roughness [m]: describes effective bed roughness
- shear stress model: type of model used for calculating wind shear stress in wave formation

Wave: Open water surface waves are surface waves which occur at the surface of an open water body. That is, a wave that is guided along the interface between water and air. As the wind blows, pressure and friction forces associated with the wind perturb the equilibrium of the ocean surface. The wind actually transfers some of its energy into the water. The water is able to gain energy from the wind because of the friction between the wind and the water. This causes the surface particles to move in an elliptical motion, which is a combination of longitudinal (back and forth) and transverse (up and down) wave motions.

[Source: Modified from Wikipedia (www.wikipedia.org)]

Erosion:



- doerosion: if selected calculate erosion
- Huttula / Bottom stress / Hungarian: erosion formula type
- eroding variable: select model variable for which erosion is calculated
- cr. shear velocity [cm/s]: critical bottom layer velocity below which there is no erosion; u_{cr}
- erosion speed [g m^{-2} d^{-1}]: base magnitude of erosion material flux per square meter and day; c_{sp}

- exponent 1 (c_1), exponent 2 (c_2), Hungarian 1 (h_1), Hungarian 2 (h_2), Hungarian 3 (h_3): parameter values for the different erosion formulas

The erosion rate formula by Huttula is of the form:

$$M = c_{sp}(u^{c_2} - u_{cr}^{c_2})^{c_1}, u > u_{cr}$$

$$M = 0, u \leqslant u_{cr}$$

Often $c_1 = c_2 = 1$. Simple bottom stress dependent erosion rate formula is:

$$M = c_{sp}\tau^{c_1}, u > u_{cr}$$

$$M = 0, u \leqslant u_{cr}$$

Hungarian formula is:

$$M = c_{sp}(\tau - \tau_{cr})^{c_1}, \tau > \tau_{cr}$$

$$M = 0, \tau \leqslant \tau_{cr}$$

In addition Hungarian formulation specifies settling rate:

$$M_{set} = \frac{C}{\Delta z} h_1 (1 - \frac{\tau}{h_3})^{h_2}$$

where C is bottom layer concentration and Δz bottom layer thickness.

Other bed erosion related model options are bed load and cohesive sediment models. These are obtainable by setting them active in the source code. WUP-FIN Model Report discusses in detail about sediment modelling including bed erosion, cohesive sediments, bed load and bank erosion.

9.3 Landuse parameters

The model is using land use map to describe the density, height and oxygen consumption of vegetation. The model is using that information to calculate the vegetation impact on the wind, flow friction, and oxygen levels. The land use map needs to be first imported to the model application prior to set-up the parameters for each landuse class.

The land use types affect both the hydrodynamic and water quality parameters. In the hydrodynamic model the effect comes from three sources:

- wind fetch
- wind shielding
- vegetation stress (friction)

> The model can be run without the landuse map. The flow friction can be set up for different areas of model area also by defining that in the physical parameters window (see Section 9.2.3 – Friction coefficients).

Land cover is the actual distribution of vegetation, water, desert, ice, and other physical features of the land, including those created by human activities.

Land use characterizes the human use of a land cover type. Forests, for instance could be used by foresters (selective logging), rubber tapers, or not at all.

In the model the land use map is used mainly for defining the physical characteristics of the vegetation and thus, the land use map here should be understood more as land cover map.

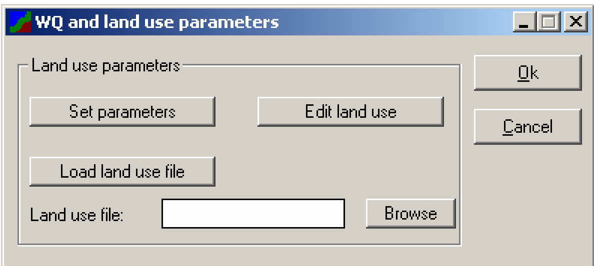
9.3.1 Importing land use map to the model application

The land use map should be in *.ipd format. This is possible to do from other grid format by using RLGis program. Once you have the land use file covering the area you are modelling, open it in RLGis and save the file again to *.ipd file. More about RLGis programme can be found from Appendixes (see **RLGis manual**).

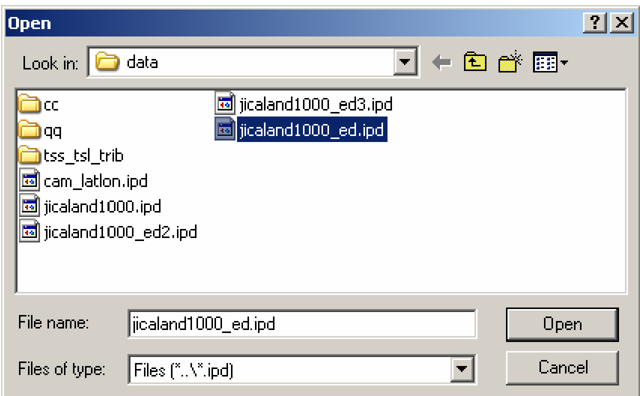
The land use map you are using should cover the whole model application area. It can exceed it but it should be in the same coordinate system than the model application (to define the grid coordinate see Section 9.1 on page 120).

To import the land use map follow the steps below:

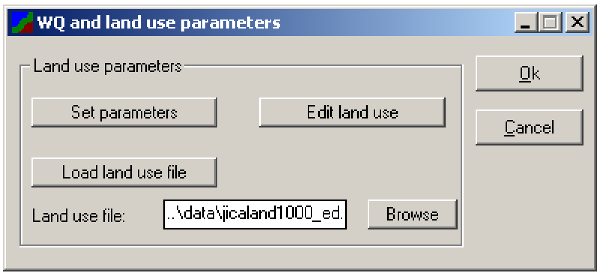
1. In 3D flow model select **Model – Land use related parameters**… and following window appear:



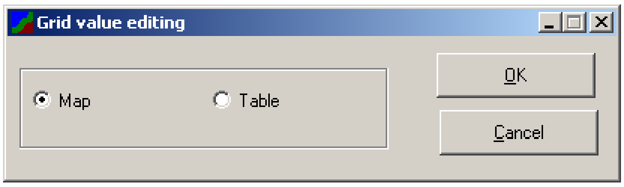
2. Browse the land use file: click **Browse** and select the right file (here example from Tonle Sap model application)



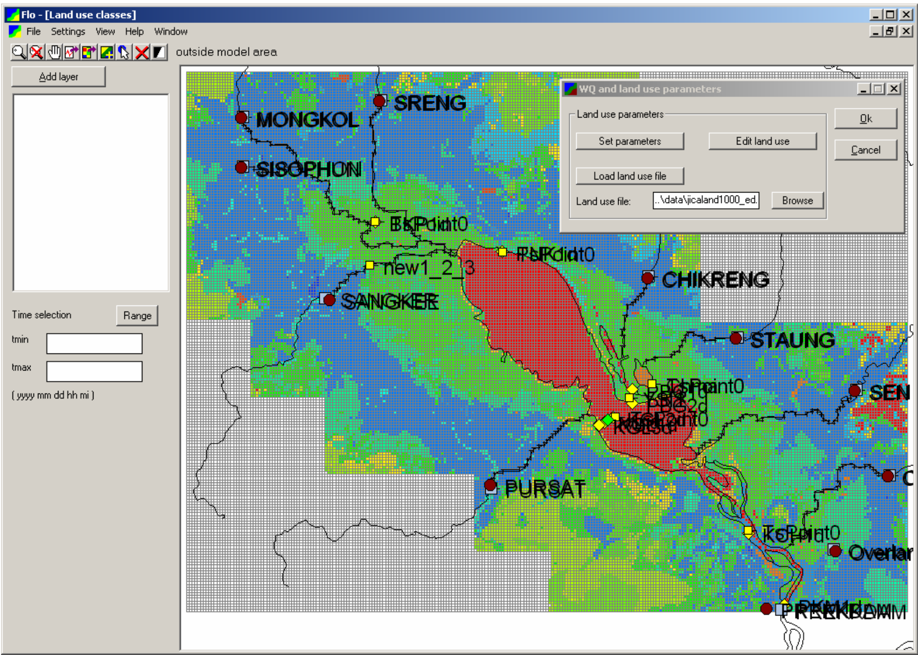
3. Click **Open** and you'll see the land use file path in the window



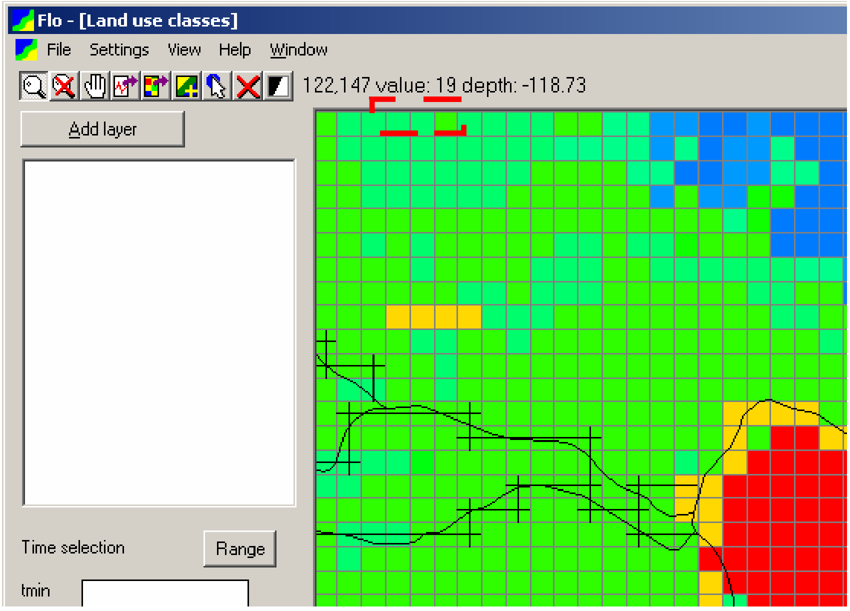
4. Press **Load land use file** to load the landuse area corresponding the application area for model to use.
5. You can see and modify the land use data by clicking **Edit land use**…
6. From the Grid value editing window select either Map or Table, depending on whether you want to see/edit the data in map or table format. Then click **OK**.



7. **Map**: to see the landuse map in map format, select **Map** in the Grid value editing window and press **OK**. The following map window appears.



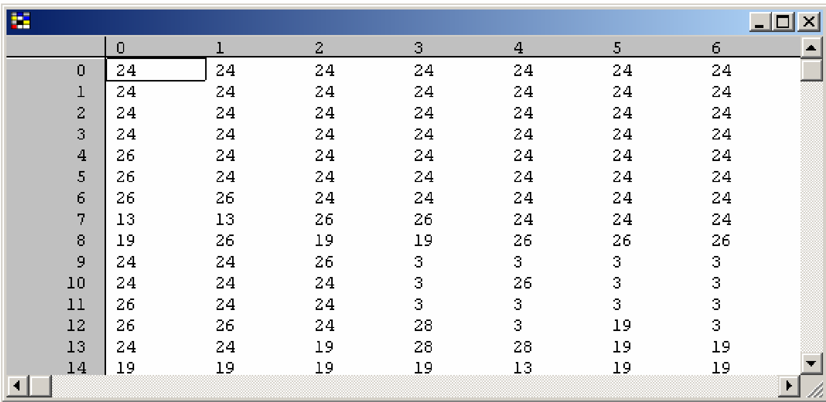
   a. User can manage the map as the main model window by using the tools in the toolbar
   b. User can see the landuse value for the cell, where your mouse pointer is, on the next to the toolbar.



c. See Section 9.3.3 – Modify land use map to learn how to edit the landuse values.

8. **Table**: to see the LU values in table format, select **Table** in the Grid value editing window and press **OK**. The following table appears.

   a. Every grid cell has one value in the matrix. User can locate the grid in the map by using the grid coordinates to calculate the place in matrix. The (0, 0) value in the depth matrix is the upper right corner in the grid.



	0	1	2	3	4	5	6
0	24	24	24	24	24	24	24
1	24	24	24	24	24	24	24
2	24	24	24	24	24	24	24
3	24	24	24	24	24	24	24
4	26	24	24	24	24	24	24
5	26	24	24	24	24	24	24
6	26	26	24	24	24	24	24
7	13	13	26	26	24	24	24
8	19	26	19	19	26	26	26
9	24	24	26	3	3	3	3
10	24	24	24	3	26	3	3
11	26	24	24	3	3	3	3
12	26	26	24	28	3	19	3
13	24	24	19	28	28	19	19
14	19	19	19	19	13	19	19

   b. See Section 9.3.3 – Modify land use map to learn how to edit the landuse values.

9.3.2 Land use parameters

The land use types affect both the hydrodynamic and water quality parameters. In the hydrodynamic model the effect comes from three sources:

- wind fetch
- wind shielding
- vegetation stress (friction)

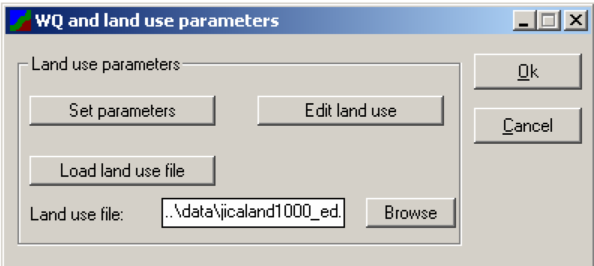
Average vegetation height, cover and friction are given for each land use type. These in turn determine wind and flow friction in different depth zones. Wind friction is diminished proportional to the vegetation cover. Vegetation friction affects flow only in the layers that lie are lower or equal height with vegetation.

The user can define following parameters for each landuse class or group of landuse classes:

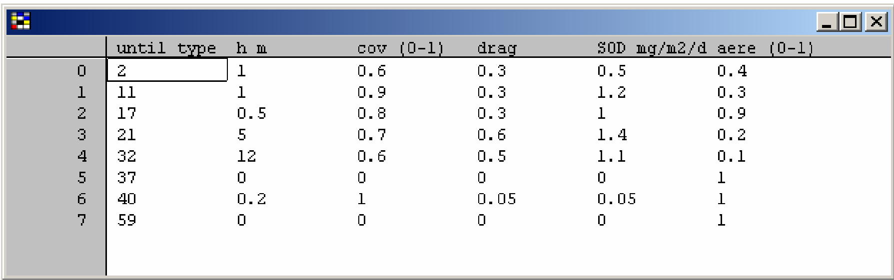
- h [m]: characteristic height of the vegetation
- cov (0-1): vegetation coverage (what ratio of the surface is covered by vegetation)
- drag: vegetation friction (drag) coefficient; in the bottom layer added to the bottom friction coefficient; in Tonle Sap typical values are 0.3 – 0.6, that is 10 – 20 times higher than the basic linear bottom friction coefficient that has value 0.03.
- SOD [$mg/m^2/d$]: sediment oxygen demand; in Tonle Sap typical values are 0.5 – 1.4.
- Aere (0-1): aeration scaling coefficient

Follow the steps below to edit/set LU parameters:

1. Select Model – Land use related parameters… from the main menu and the following window appears



2. User can edit the LU parameters by clicking Set parameter in WQ and LU parameters window. The following table appears (number of LU types depends on the application).



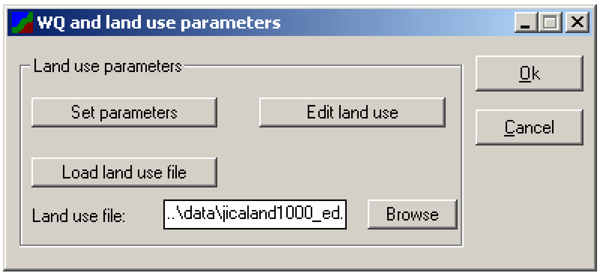
	until type	h m	cov (0-1)	drag	SOD mg/m2/d	aere (0-1)
0	2	1	0.6	0.3	0.5	0.4
1	11	1	0.9	0.3	1.2	0.3
2	17	0.5	0.8	0.3	1	0.9
3	21	5	0.7	0.6	1.4	0.2
4	32	12	0.6	0.5	1.1	0.1
5	37	0	0	0	0	1
6	40	0.2	1	0.05	0.05	1
7	59	0	0	0	0	1

3. User can edit existing data by clicking the number and typing the new one. Similar technique needs to be used when inserting new data.
4. When you have finished the edits, press **Store** in the toolbar to save the made changes. If you want to cancel the changes, press **Cancel**.

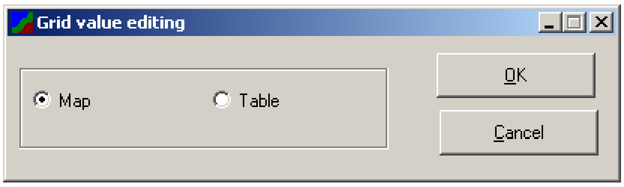
9.3.3 Modify land use map

User has two options to modify the LU values, either through map or table based interface. Both of the options are explained in more details below

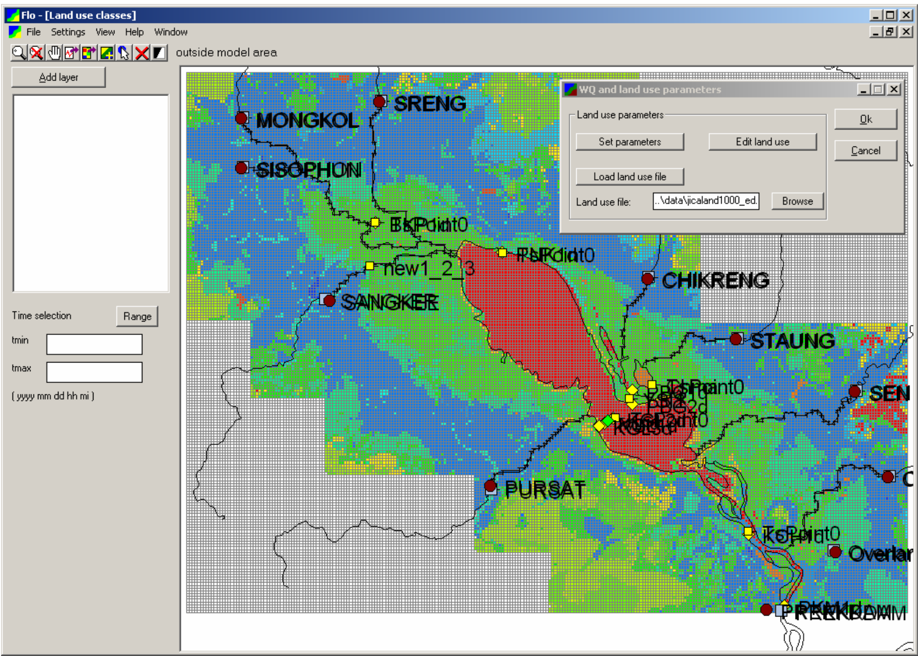
1. Select Model – Land use related parameters… from the main menu and the following window appears



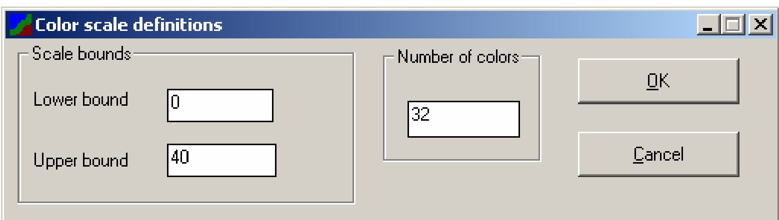
2. Select **Edit land use**



3. From the Grid value editing window select either Map or Table, depending on whether you want to see/edit the data in map or table format. Then click **OK**.
4. **Map:** to edit the LU values in map format, select **Map** in the Grid value editing window and press **OK**. The following map window appears.

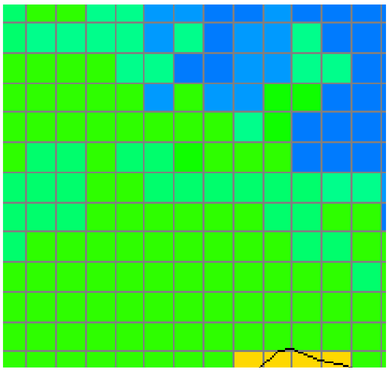


   a. User can manage the map as the main model window by using the tools in the toolbar
   b. If the colour scale is not right, set up the colour scale by selecting **View – Colour scale…** from the main menu.

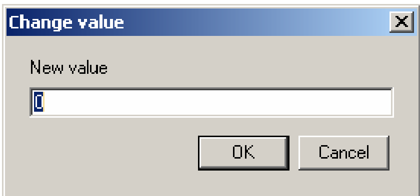


c. The lower bound LU classes are shown in blue colour and higher with red one

d. Zoom in to the area you want to edit



e. Select either continuous or single mode from the toolbar to modify the grid depths. With the **Continuous mode** the user can enter the value for the LU value and then continuously edit the channels. With the **Single mode** the user needs to define every time separately the value for the LU value.



f. now you can modify the grid depths by clicking or dragging the selected grid cells by left mouse button

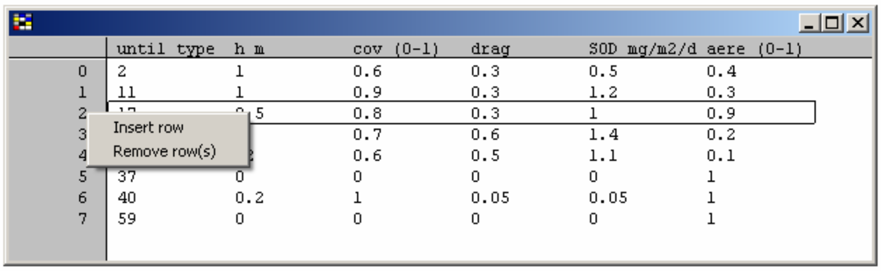
g. when you have finished the editing, close the Grid depths map window and the modifications will be stored

5. **Table**: to edit the LU values in table format, select **Table** in the Grid value editing window and press **OK**. The following table appears (number of LU types depends on the application).

	until type	h m	cov (0-1)	drag	SOD mg/m2/d	aere (0-1)
0	2	1	0.6	0.3	0.5	0.4
1	11	1	0.9	0.3	1.2	0.3
2	17	0.5	0.8	0.3	1	0.9
3	21	5	0.7	0.6	1.4	0.2
4	32	12	0.6	0.5	1.1	0.1
5	37	0	0	0	0	1
6	40	0.2	1	0.05	0.05	1
7	59	0	0	0	0	1

a. Every grid cell has one value in the matrix. User can locate the grid in the map by using the grid coordinates to calculate the place in matrix. The (0, 0) value in the depth matrix is the upper right corner in the grid.

b. User can delete or add layers by selecting first the row to delete or to where to add new by left mouse button. By clicking the right mouse button in the row number you get the menu to insert or remove row(s).



c. User can edit existing data by clicking the number and typing the new one. Similar technique needs to be used when inserting new data.

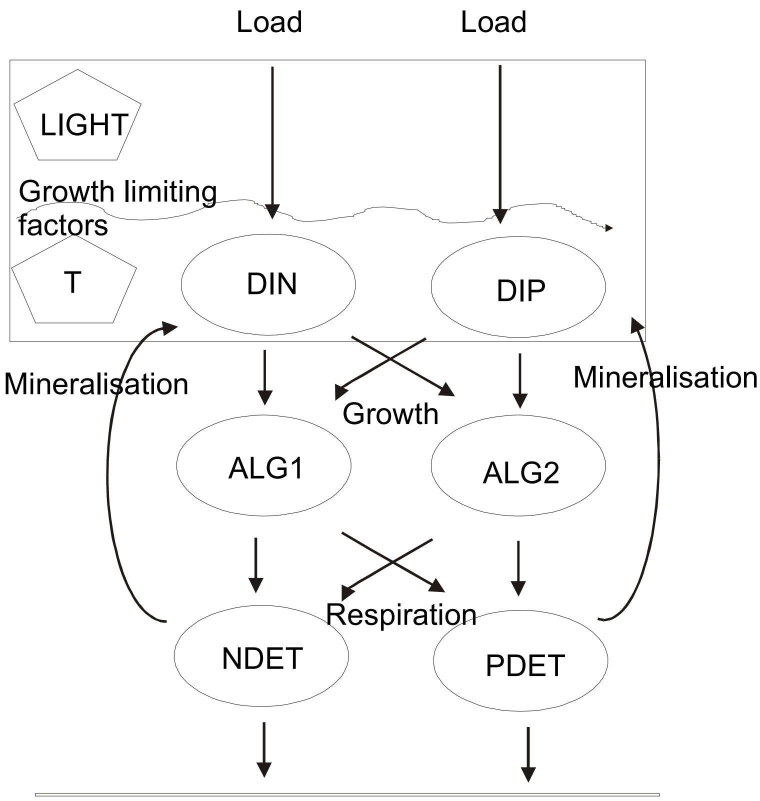
d. When you have finished the edits, press **Store** in the toolbar to save the made changes. If you want to cancel the changes, press **Cancel**.

9.4 Water quality parameters

In this EIA 3D model only few water quality parameter computation procedures have been attached to the model. There exists a separate water quality model which can be combined to the EIA 3D hydrodynamic model and optimised to compute long time periods, large number of variables and complex processes. As an example one of the primary productivity models is presented below. The calculation variables are:

- ALG1 phytoplankton biomass, group 1 (mg/l)
- ALG2 phytoplankton biomass, group 2 (mg/l)
- FIL_A filamentous algae biomass, group A (mg/m^2)
- FIL_B filamentous algae biomass, group B (mg/m^2)
- DIPP concentration of dissolved inorganic phosphorus (µg/l)
- DINN concentration of dissolved inorganic nitrogen (µg/l)
- PDET detritus nitrogen concentration (mg/l)
- NDET detritus nitrogen concentration (mg/l)

Corresponding processes are presented in the figure below.



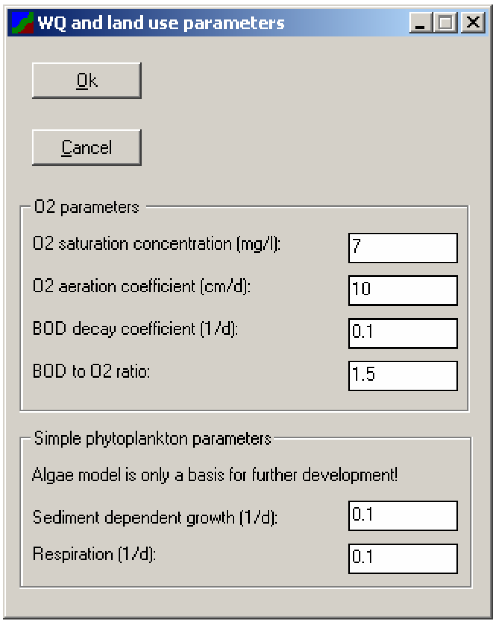
Compared to the dedicated water quality model WUP-FIN Mekong 3D provides relatively limited set of processes. This has been however augmented in year 2008 by development of a primary productivity model for Mekong and especially for Tonle Sap. It includes terrestrial macrophytes (floodplain plants), macrophytes (primary producers attached to solid surfaces) and phytoplankton (floating primary producers). The new model development is presented in a separate report.

In the Mekong applications dissolved oxygen concentrations are governed by sediment oxygen demand, BOD (Biochemical Oxygen Demand) and aeration. The equation for oxygen without transport and diffusion is:

$$\frac{\partial c}{\partial t} = -k_1 BOD + k_2 (c_s - c)$$

Here k_1 is the BOD ratio (how much oxygen each decaying BOD unit consumes), k_2 aeration rate and c_s oxygen saturation value. SOD (Sediment Oxygen Demand) needs to be added to the equation in the bottom layer. SOD depends strongly on land use/ vegetation type and is given with other land use dependent parameters.

The aeration and bottom sediment oxygen demand values are given for each land use class. It is obvious that the vegetation cover and land use type have a strong effect on aeration and the decaying biological material on the ground. The degradable biological material in the water phase (BOD) is transported around and consumes oxygen with a rate that is not assumed to be strongly land use dependent.The basic 3D model water quality parameters can be set by selecting **Model – Water quality parameters…** from main menu:



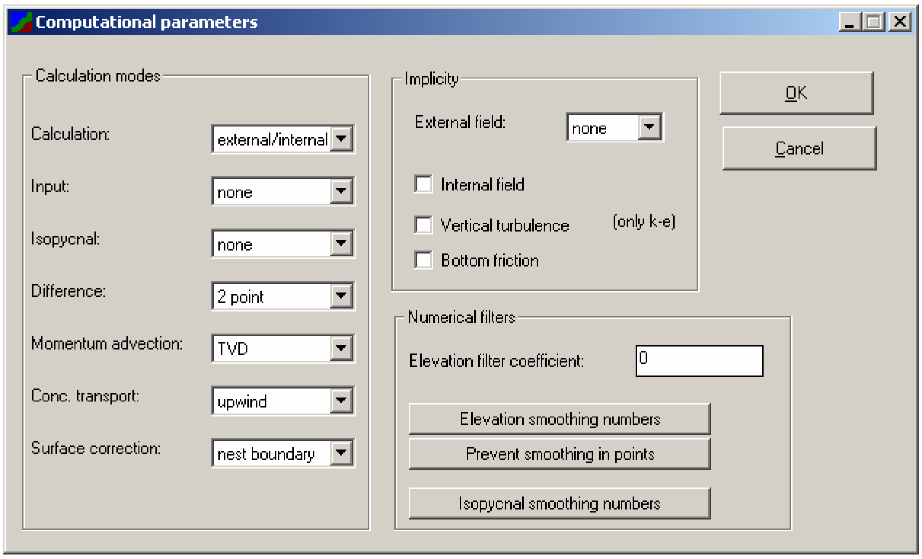
The parameters are:

- Dissolved oxygen (O_2 or DO)
    - O2 saturation concentration (mg/l): maximum dissolved oxygen concentration; saturation concentration depends on temperature; in the simplified model temperature variations are assumed not to impact appriaciably saturation concentration (c_s in the above equation), value about 7 mg/l
    - O2 aeration coefficient (cm/d): is function of wind speed, but here constant value is used (k_2 in the above equation), value about 10 cm/d
- BOD (Biological Oxygen Demand, represents oxidation of organic material)
    - BOD decay coefficient (1/d), value about 0.1
    - BOD to O2 ration: k_1 in the above equation, value about 1.5
- Phytoplankton
    - Sediment dependent growth (1/d): not valid, see productivity model documentation for new formulation
    - Respiration (1/d): not valid, see productivity mdoel documentation

9.5 Computational parameters

By selecting **Model – Computational parameters…** from the main menu, the user is able to set the computational parameters. The **Computational parameters** window has been divided to three sub-categories:

- Calculation modes
- Implicity
- Numerical filters



The options and their descriptions for each **Calculation mode** are presented in the following table:

Mode	**Options**	**Description**
Calculation		
	External/internal	Calculate layer velocities from 2D fields and velocity differences between layers – usually faster although not as accurate as Layer-option (see Model Report).
	Superponated	Use pre-calculated stationary flow fields for specified winds, discharges and water levels; flow fields are superponated for any specific wind, discharge and water level situation
	Read	Read set of stationary flow fields and use them
	Layer	Calculate layer velocities directly; more accurate especially for viscosity and stratification but slower

Mode	Options	Description
Input		Use pre-calculated dynamic (changing) flow fields; eliminates flow calculation making water quality and sediment calculation much faster
	None	Don't use pre-calculated flows
	Aver. asc	Averaged fields over specified timestep, Ascii-format
	Inst. asc	Instantenous flow fields stored with specified timestep
	Aver. bin	Averaged fields, binary format; **use this** for concentration simulation because it conserves water volumes; binary format stores values more efficiently than Ascii
	Inst. bin	Instantaneous flow fields, binary format
	Incr. bin	Only flow changes compared to previous timestep are stored
Isopycnal		Isopycnal simulation (see above for definition and discussion)
	None	Don't use
	Within layers	Allow interface movement only within each grid layer
	Between layers	Allow interface movement also over grid layer boundaries
Difference		
	2 point	In momentum equation each pressure gradient is calculated with 2 points (velocity components are calculated in the middle of grid cells and pressure in the corners; diagonal velocity components are calculated; requires regular grid)
	4 point	In irregular grids use all 4 corner points for pressure gradient calculation

Mode	Options	Description
Momentum advection		
	1 upwind x/y	Upwind-algorithm in x- and y-directions; quite accurate and fast
	centered	Centered algorithm; usually not stable because of non-physical characteristics
	TVD	TVD (Total Variation Diminishing)-algorithm; reduces numerical diffusion; precise but very time consuming; use for highly non-linear problems such as dam-break scenarios
	1 upwind 45	Upwind-algorithm in diagonal direction; experimental
	2 upwind 45	Second upwind-algorithm in diagonal direction; experimental
	2 upwind x/y	Second updwind-algorithm in x- and y-directions; experimental
Conc. transport		Concentration transport
	Upwind	Upwind-algorithm
	Flux	Flux-corrected anti-diffusion in the vertical direction
	TVD	TVD-algorithm in the horizontal direction
Surface correction		In case of non-conservative (not maintaining water volumes) nesting correct volumes
	Nest boundary	Divide correction to the boundary
	All	Dividie correction to all cells

The **implicity** options are defined in the following table:

Mode	Options	Description
External field		Use implicity for 2D field calculation; works only with external/internal mode (see above); should not be used when water level changes are large, such as in Mekong
	None	Don't use
	Mode 1	Implicity algorithm 1, least stable
	Mode 2	Implicity algorithm 2, more stable
	Mode 3	Implicity algorithm 3, most stable
Internal field		Necessary for high velocity, friction and viscosity cases; k-e turbulence models in general requires implicity; keeps solution stable but too large internal time step can lead to non-physical solutions such as articifical eddies or artifically high velocities

Mode	Options	Description
Vertical turbulence (only k-e)		Implicit calculation in k-e turbulence model
Bottom friction		Implicit bottom friction term; can stabilise solution with high friction

Explicit methods calculate the state of a system at a later time from the state of the system at the current time, while an **implicit method** finds it by solving an equation involving both the current state of the system and the later one. Mathematically, if $Y(t)$ is the current system state and $Y(t + \Delta t)$ is the state at the later time (Δt is a small time step), then, for an explicit method

$$Y(t+\Delta t)=F(Y(t))$$

while for an implicit method one solves an equation

$$G(Y(t),Y(t+\Delta t))=0$$

to find $Y(t + \Delta t)$.

[Source: Wikipedia (www.wikipedia.org)]

The **Numerical filters** sub-category includes the following options:

- Elevation smoothing numbers: nested grid dependent surface elevation smoothing number; number tells into how many surrounding cells water amount changes are divided (1 = nearest, 2 = nearest and second nearest etc.); further away surrounding points are less they receive/ contribute water; adds implicity into the model and enables longer external time step; when integer value approaches grid dimensions, dynamic solution approaches implicit solution (see definition of implicity below); can be given also with **Edit nest data** in the grid specification dialog window (see above), but is more accessible here
- Prevent smoothing on points: prescribe specific points such as narrow structures where smoothing is not applicable
- Isopycnal smoothing numbers: similar smoothing numbers for isopycnal calculation

In previous user interface versions also diffusive elevation filter coefficient could be prescribed. This can be understood as surface elevation diffusion or increasing water surface viscosity. The problem with this is formulation is that calculation is not conservative (maintaining water volumes). At the moment this option has been elminated. In the model control file it is still possible to prescribe the coefficient as well similar coefficient for surface *change*.

Remember to activate and set up the time steps for each required process activated in **Computational parameters** window

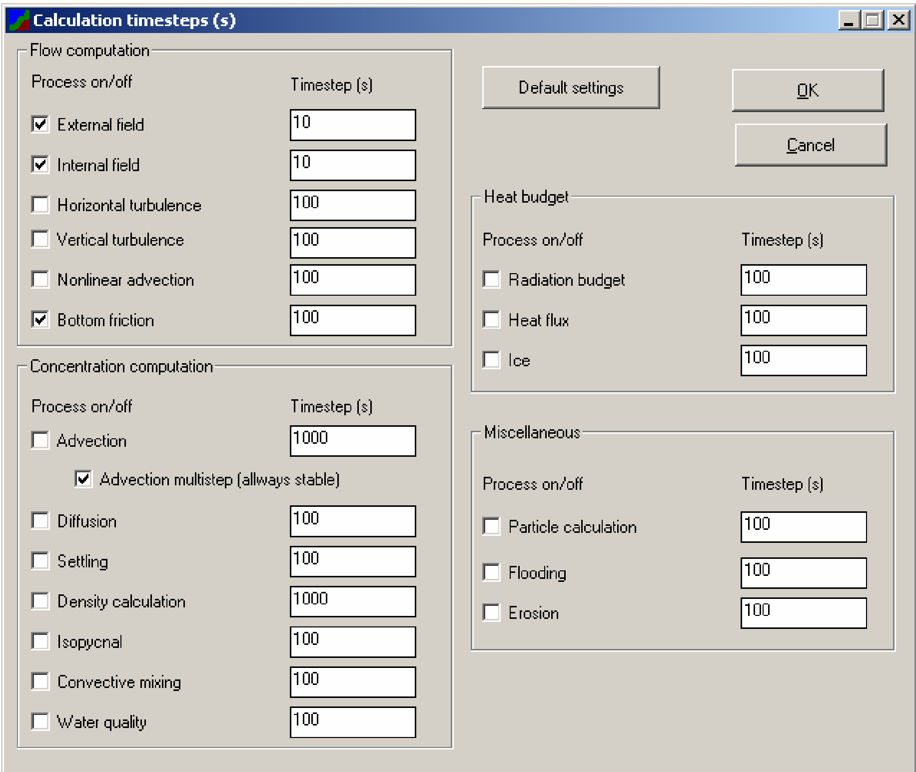
9.6 TIME STEPS

User is able to define separate time steps for each computation process as illustrated below. For each time step there is a check box under the **Process on/off** column where user can check the process will be computed or not. The second column, Timestep (s), defines the time step for process in question.

The **Calculation timesteps (s)** window is divided to four sub-categories:

- Flow computation
- Concentration computation
- Heat budget
- Miscellaneous

The **Calculation timesteps** window accessible through **Model – Timesteps** is illustrated below



The description for **Flow computation** processes are given below:

Process	Description
External field	If computation model is External/internal (see previous chapter), calculates 2D fields; if mode is Layer, calculates basic flow; allways on
Internal field	If computation model is External/internal, calculates internal velocity distribution (=velocity differences between layers); if model is Externan/internal and Internal field is not calculated flow field is homogenous in vertical direction (equals to situation where vertical viscosity is infinite); if mode is Layer, doesn't have any impact;
Horizontal turbulence	Horisontal turbulence is calculated with Smagorinsky model
Vertical turbulence	Vertical turbulence calculated with alternative models
Nonlinear advection	Momentum advection calculated
Bottom friction	Bottom friction computation, usually always on

The description for **Concentration computation** processes are given below:

Process	Description
Advection	Concentration advection
Advection multistep	If on model cheks advection stability and divides advecetion timestep into smaller parts resulting in inner iterations; within inner iterations flows remain fixed
Diffusion	Concentration variable vertical diffusion. In most cases not used to compensate for numerical diffusion.
Settling	Concentration settling. Used if sedimentation, nutrient settling with dead phytoplankton or similar processes are calculated.
Density calculation	Density variation effect to flow, used if salinity, temperature or large amounts of dissolved substance is included into the model.
Isopycnal	Calculate isopycnal stratification (see above). Can be calculated together with normal temperature and salinity computation.
Convective mixing	Mixing of water masses in case of reversed stratification (heavier water on top of lighter one)
Water quality	Computation of water quality processes (aeration, BOD decay etc.)

The description for **Heat budget** processes are given below:

Process	Description
Radiation budget	Computation of incoming solar and outgoing long wave radiation
Heat flux	Computation of water surface heat exchange
Ice	Computation of ice formation and melting

The description for **Miscellaneous** processes are given below:

Process	Description
Particle calculation	Lagrangian (particle) calculation
Flooding	Flood mode of calculation for large water elevation changes and wetting and drying; used for all Mekong applications; if this is not on flow depth changes are assumed to be small relatively to the intial depths; flooding time step is very time consuming because active computation domain and water depths change resulting need to update most of model tables
Erosion	Erosion calculation

Guidelines for setting-up timesteps are:

- External field (2D) and direct layer velocity calculation is governed by gravitational wave propagation speed. External field explicit method stability criterion is given by Courant-Friedrichs-Levy (CFL) computational stability condition:

  $$\Delta t \leqslant \frac{\Delta x \Delta y}{2\sqrt{g H_{max}} + u_{max}} \sqrt{\frac{1}{\Delta x^2} + \frac{1}{\Delta y^2}}$$

  where g is gravitational constant, H_{max} maximum depth and u_{max} maximum velocity. When velocities are small and grid is regular this simplifies to:

  $$\Delta t \leqslant \frac{\Delta x}{\sqrt{2 g H_{max}}}$$

  Because mdoel calculation system is diagonal, the effective grid length is $\sqrt{2}$ times longer than the basic grid length, and the stability criterion is:

  $$\Delta t \leqslant \frac{\Delta x}{\sqrt{g H_{max}}}$$

- Internal field stability can be given analogously dependent on internal gravity wave speed (largest speed around 2 m/s) and maximum *advective* speed. This criterion is much less restrictive than the external one given above. However, with large viscosity internal time step (and external time step in layer-mode) may need to be decreased. Especially application of k-e turbulence model results often in high vertical viscosity and need for implicit calculation and small time steps. Approximate explicit stability condition for internal time step is:

$$\Delta t \leqslant \frac{\Delta z^2}{2D}$$

where D is the viscosity coefficient. Typical vertical grid size is on the order of 1 m and higher vertical viscosity values can easily reach 0.1 m^2/s. This gives, for *explicit* computation, 5 s stable time step. For more precise formulation see Diploma Thesis by J. Koponen.

- When implicity is used, solution may be stable although result is non-physical. In this case time step needs to be reduced. See further discussion above.
- Algorithm for momentum advection calculation has required homogenisation of external, internal and momentum advection timesteps (that is best results have been obtained when same timeste has been used for these processes). New algorithm not yet implemented in the Mekong applications relaxes this restriction.
- Bottom friction calculation can become restrictive with high flow velocities especially in the nonlinear friction case. Then the stability criteria is:

$$\Delta t \leqslant \frac{\Delta z^2}{2D}$$

- Use rather large advection timestep (e.g. 3600 s) and **Advection multistep** for concentration calculation to quarantee stability. Unstable calculation, large flow and small grid cell size near bottom or shore may result in very restrictive time step. In this case stabilise flow calculation with smaller time steps or specify minimum grid cell volume in **Model/ Grid data/ minvol.** You can also uncheck **use real volumes** in **Model/ Grid data** to increase concentration grid cell size near shores and channel areas.
- In general other time steps can be kept rather large, on the order of 1'000 s.
- Flooding time step should be as large as possible because it is very time consuming; on the other hand too large step can result in oscillations in the flow solution. Usually time steps between 1'000 and 10'000 s can be used.

Examples of stable external (2D field) time steps with different grid sizes and maximum depths

Grid size **Max depth**	**50 m**	**100 m**	**500 m**	**1000 m**
5 m	7	14	71	143
10 m	5	10	50	101
15 m	4	8	41	82
20 m	4	7	36	71

Simple rules for setting-up model timesteps are:

- Give external/ layer time step by

$$\Delta t \leqslant \frac{\Delta x}{\sqrt{g H_{max}}}$$

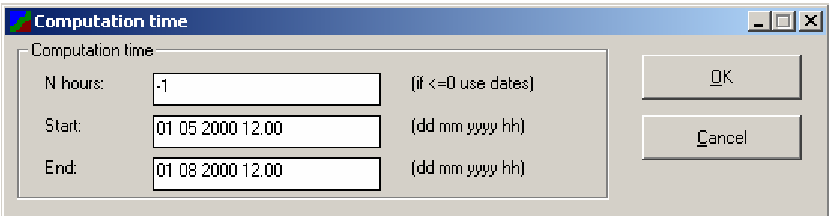
- You can use surface smoothing and external implicity to increase time step. However, in case of large water level changes such as encountered in the Mekong, don't use these options.
- Use implicity for internal mode, vertical turbulence and and bottom friction (in **Model/ Computational parameters**).
- In case of External/internal mode calculation, increase internal times step keeping calculation stable and physical.
- In case of momentum advection keep external, internal and momentum advection time steps same (to be changed in next model version).
- Bottom friction time step can be kept the same as external one.
- Use **Advection multistep** for concentration calculation.
- In general other time steps can be kept rather large, on the order of 1'000 s.

User needs to optimise the time steps for each application separately. If the model is unstable, check the time steps and try to decrease them. See Chapter 18 – Troubleshooting for more information.

9.7 Computation period

Computation period definition (**Model – Computation period…**). Same data as in the **Model/Run**… window (see Section 12.1 Run the model).

The **Computation time** window is presented below



Computation time

N hours - number of hours to compute, if less than zero then use dates below

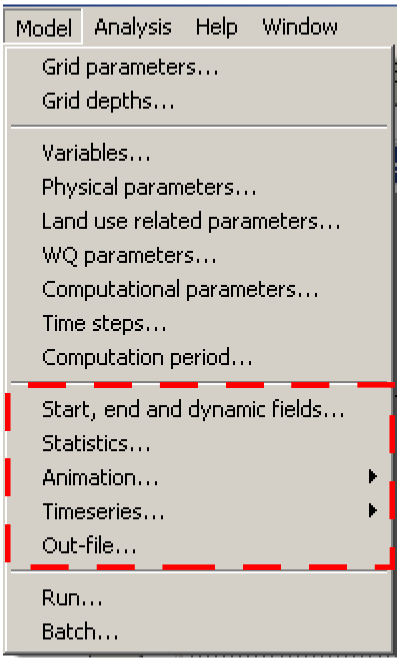
Start - computation start time in format 'dd mm yy hh'

End - computation end time in format 'dd mm yy hh'

Press **OK** to apply the changes, **Cancel** to return to main window without changes made to the **Computation time** settings.

Remember to put value <=0 to the N hours if you want to use the start and end dates to define the computation time.

10 OUTPUT



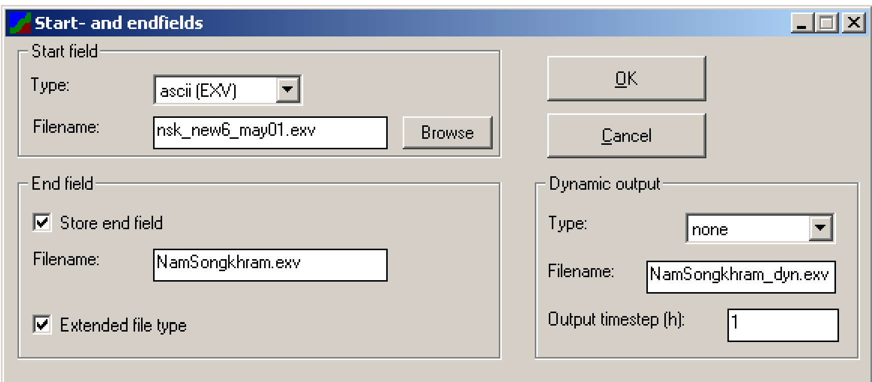
This chapter describes the model output options and settings

The chapter is divided to four parts:

10.1 Start, end and dynamic fields

Settings for model initial and end concentration files. The model initial concentration fields can be read from a file. The initial concentration file must be an end situation file saved previously. If the concentrations are not read from a file then the initial values defined for each variable are used to initialise the concentration field (see Section 9.1 Variables).

To open the “Start and end fields” window, go to **Model – Start, end and dynamic fields…** and the window opens



Start field

Type – Select some of the data types (ascii (EXV), aver. bin, inst. bin or incr. bin) to use the start field. If selected None, no start field will be used. The following file types are used

- Ascii (EXV):
- Aver. bin:

- Inst. bin:
- Incr. bin:

See chapter 9.5 for the definitions of the file types.

Filename - defines the initial concentrations file name. This must be a file previously saved as an endfield (see next section).

End field

Store end field – check this if you want to store the end field. Usually checked.

Filename – filename for the end field. You can use the file as start situation for other simulations if computational variables are same in both simulations

Extended file type – leave this checked allways. Provided for compatibility with older applications.

Dynamic output

Type – Select one of the data types listed below with the description to store the dynamic output. If selected None, no dynamic output will be stored. The available data types are

- Aver. asc
- Inst. asc
- Matrix asc
- GRIB
- Aver. bin
- Inst. bin
- Incr. bin
- GRADS
- BIL

For explanations of Aver, Inst and Incr types see chapter 9.5. Matrix asc, GRIB, GRADS and BIL types are used for provision of flow fields for external applications. Matrix Ascii format provides possibity to provide flow, water depth, concentration and other data to for instance spreadsheet software. GRIB is standard meteorological data format. GRADS and BIL are GIS-formats. GRIB and GRADS formats are not functional in the current model version.

Filename - defines the file name for the dynamic output

Output timestep (h): defines the timestep how often the dynamic output will be stored.

Some of the dynamic output files can grow quite large (>100MB). Thus, make sure that you have enough space in your hard drive. Test the timestep as well that it suites to your purposes.

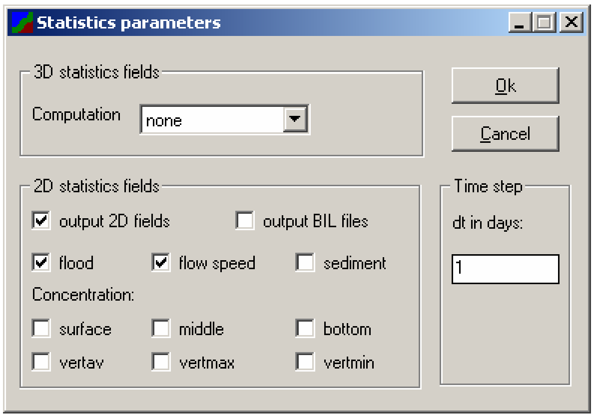
10.2 Statistics parameters

User is able to define the stored statistic parameters for the results in "**Statistics parameters**" window (**Models – Statistics…**). The window is divided to three sub-categories:

- 3D statistics fields

- 2D statistics fields
- Time step

The following window appears for the parameter settings:



3D statistic fields

Output of 3D fields of computational variable statistics. The format of the output is Exv. asc (see chapter 9.5). The following computation options can be selected by the user:

- None: 3D statistics are not calculated
- Basic: averages of computational variables
- Extended: averages of computational variables + additional characteristics such as flow average speed, percentage of time exceeding specified threshold speed, maximum temperature, surface elevation variability etc. Specific outputs tailored in model code.

As the names for statistical characteristics don't necessarily conform to standard model names, it may be required that for proper output the names need to be edited in the ouput files.

2D statistics fields

Output of 2D fields of computational variable statistics. The format of the output is Exv. asc (see chapter 9.5). User has following options when dealing with the 2D statistics fields:

- Output 2D fields: User has to first select whether the 2D output will be stored during the calculation or not. If this is needed, then the 2D output check box should be checked on.
- Output BIL files: User can also save the output as BIL files which can be used after simulation in any GIS program (as ArcView, ArcGIS, DIVA, RLGis, etc)

User can select which parameters will be stored to the 2D statistics field. The selection is divided to two main classes as listed below:

- The Flow and sediment parameters
  - Flood: flood related parameters as
    - Peak Flood Time
    - Flood Duration Time
    - Flood Arrival Time
    - Flood Drying Time
    - Maximum Depth
    - Average Depth,
  - Flow speed: flow speed at three layers: bottom, middle, and surface

    - Sediment: sediment related parameters as suspended sediment concentration and bottom sedimentation
- Concentration
    - Surface: surface layer concentration
    - Middle: middle layer concentration
    - Bottom: bottom layer concentration
    - Vertav: vertical average
    - Vertmax: maximum value in the vertical
    - Vertmin: minimum value in the vertical

Time step

Defines the timestep how often the 2D statistical fields will be stored during the simulation.

10.3 ANIMATION OPTIONS

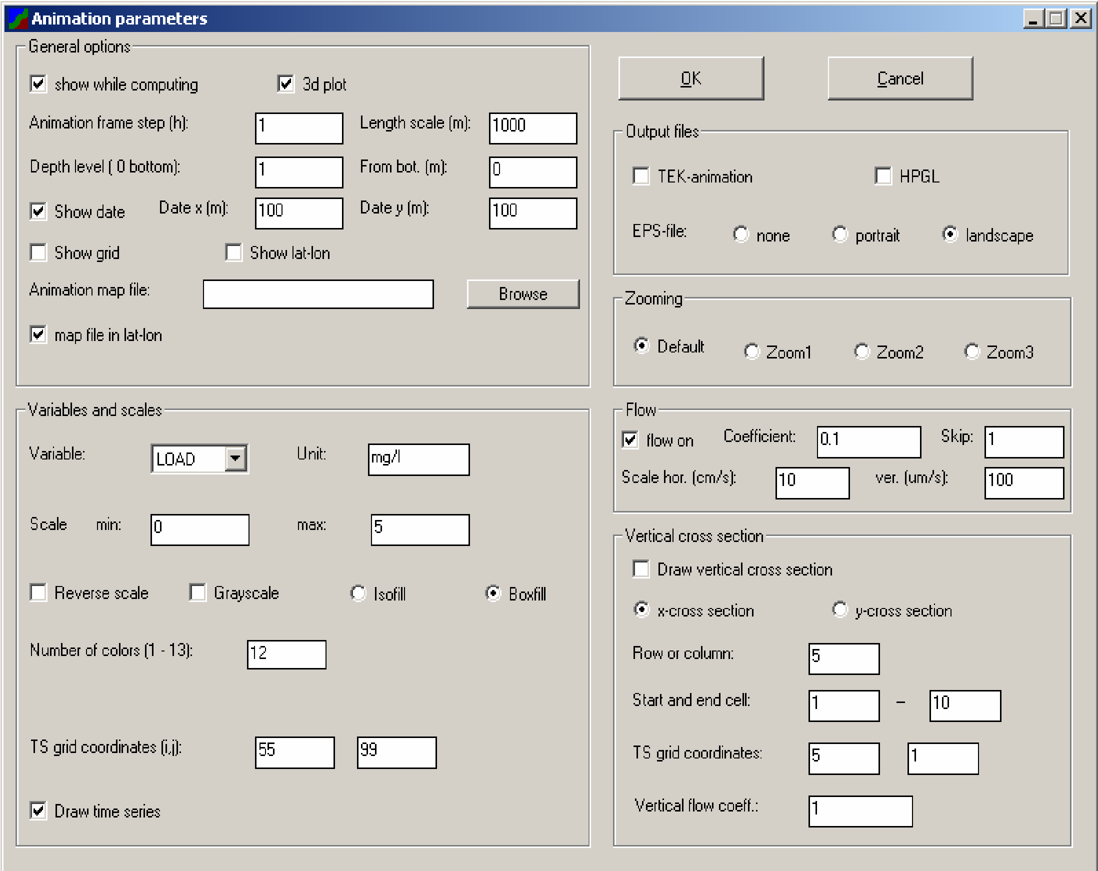
Animation picture settings can be defined here. The animation options menu is divided to three main sections as follows

- Animation options – the main animation picture settings
- Advanced animation options – more advanced and rarely used settings
- Update and delete animation ts – settings related to the animation timeserie

10.3.1 Animation options

The main animation picture settings can be defined in the **Animation parameters** window (**Model – Animation – Animation options…**). The window is presented below and is divided to six sub-categories:

- General options
- Variables and scales
- Output files
- Zooming
- Flow
- Vertical cross section



General options

Show while computing – display animation picture while computation proceeds

3d plot – display 3D plot while computation proceeds

Animation frame step (h) – Defines how often the concentration is sampled to animation. If the step is small (1h) the animation contains more information about the time evolution of the concentrations. However, a small timestep produces big animation files (usually about 24 h).

Length scale (m) – shown length scale length

Depth level (0 bottom) – animation depth level, 1 is surface and 0 is bottom, other layers 2-9 and then from a, b, c, etc depending how many layers there are in the model application

From bot. (m) – in the bottom layer output, what distance from bottom (if 0 in each grid cell middle of the bottom layer; observe that the bottom layer thickness varies!)

Show date – shows date and time of each animation frame

Date x (m): date output x-distance in m from origo

Date y (m): date output y-distance in m from origo

Show grid – show computational grid

Show lat-lon – show latitude and longitude on the grid boundaries

Animation map file – specification of map file for animation

Map file in lat-lon: if checked, map coordinates are in lat-long

Variables and scales

Variable, unit – animated variable, must be one of the computed variables. List with description of each variable can be found from Section 27.2 – Output selection symbols (in GUI and graphics files).

Scale min,max - concentration scale minimum and maximum

Output options

- Reverse scale: scale is inverted
- Grayscale: animation will use only different tones of gray
- Isofill / Boxfill - defines how the animations picture is shown, isofill usually looks better but is slower and takes more space

Number of colours (1-13) – user can define how many colours will be used in the animation picture scale

TS grid coordinates – location for the animation timeseries point

Draw timeseries – you can put one timeseries picture in the animation window. This is usually a good idea. The timeseries shows concentration of the animated variable in selected grid point.

Output files

TEK-animation – store animation in Tektronix-format; compact vector format; quality doesn't deteriorate in zooming); can be viewed either through model user interface (**Analysis/ Animation**) or separate simple viewing software (tek.exe, wtek.exe).

HPGL – HP printer language file output

EPS file – Encapsulated PostsSript file output

None: no output

Portrait: portrait orientation

Landscape: landscape orientation

Zooming

Default, Zoom 1-3 – sets the animation view to one of the predefined zoom areas (see Section 6.1 – Zoom)

Flow

Flow on – draws also flow field as arrows. In long animations consumes lot of storage space.

Flow scaling coefficient - changes flow arrow scaling. Good values depends on the application

Skip – number of flow arrows skipped in animation picture. Needed if the grid size is very small and arrows are not clear

Scale

hor. (cm/s) – scale for the horizontal velocities

ver. (um/s) – scale for the vertical velocities

Vertical cross section

Draw vertical cross section – check if you want to see the vertical cross section instead of horizontal view

X, y cross section – define whether you want to see the cross section in x-or y-direction

Row or column – if x-direction, select model grid row; if y-direction select column

Start and end cell – start and end in each row or column

TS grid coordinates – timeseries output coordinates in the cross-section grid coordinates; the latter coordinate coordinate is layer starting from surfcace layer

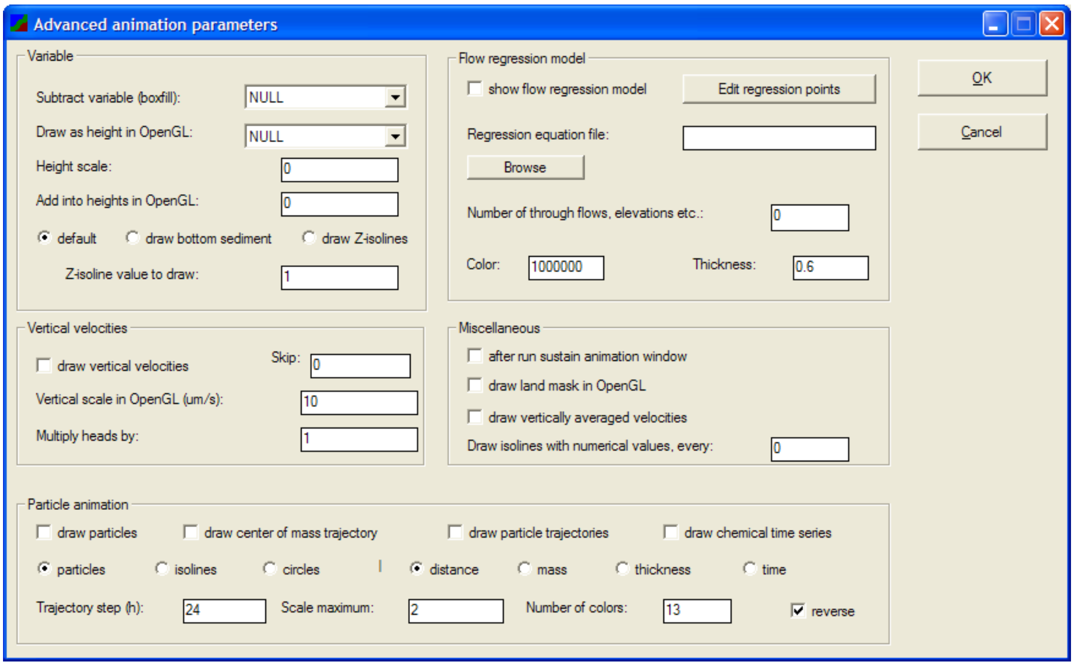
Vertical flow coefficient – scaling of the vertical velocity; normally vertical velocity is orders of magnitude lower than horisontal one.

10.3.2 Advanced animation options

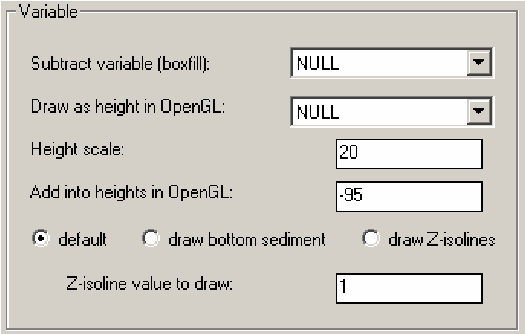
Advanced animation parameters window (presented below) is divided to five sub-categories:

- Variable
- Vertical velocities
- Flow regression model
- Miscellaneous
- Particle animation

To open the advanced animation window, select **Model – Animation – Advanced animation options…** and the following window appears.



Variable



Subtract variable (boxfill) – define which variable the colour in the OpenGL animation window (see Section 12.3 – Operating in the model computing window) represent. See list of the variables in Section 27.2 – Output selection symbols (in GUI and graphics files)

Draw as height in OpenGL – define which variable the height in the OpenGL animation window (see Section 12.3 – Operating in the model computing window) represent. See list of the variables in Section 27.2 – Output selection symbols (in GUI and graphics files)

Height scale – scale for the height

Add into heights in OpenGL – value which will be added to the water level elevations in OpenGL animation window;

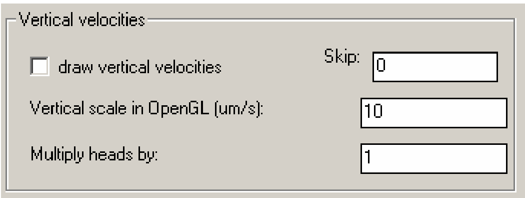
Default – draw specified variable in the water column

Draw bottom sediment – instead drawing variables in the water column, draw amount of material on the bottom (g/m^2) of the variable defined in the main animation dialogue window

Draw Z-isolines – draw height of an isoline value

Z-isoline value to draw – specify isoline value

Vertical velocities



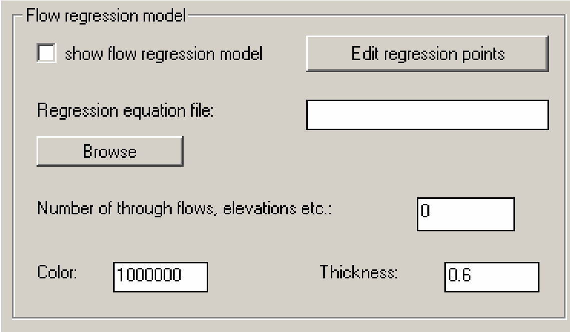
Draw vertical velocities – check this on if you want to draw vertical velocities when the vertical cross-section (see previous section) is being selected to be animated.

Skip – if the flow arrows are too dense in the animation user can select to skip every n-flow velocity arrows

Vertical scale in OpenGL (um/s) – vertical scale to be used in the animation. Depends on the application and vertical flow velocities; arrows become visible when figure is rotated

Multiply heads by – magnitude and direction of vertical flow is shown by either circles or crosses; magnitude is shown by width of the symbols and multiplication factor can be used to adjust it

Flow regression model



Show flow regression model – check this on if you want the flow regression model to be shown. Flow regression model is a statistical model derived from flow measurements linking wind, throughflow and other conditions to flow. This option will draw flow regression arrow on the animation depending on wind conditions.

Edit regression points – edit/add flow regression points

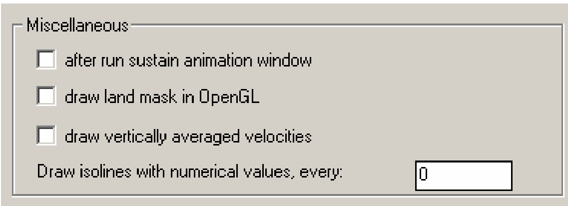
Regression equation file – define the regression equation file path by typing it to the blank space or browse it by pressing Browse.

Number of through flows, elevations, etc – number of additional parameters taken into account in the flow regression model

Colour – the colour of regression flow arrow

Thickness – thickness of the regression flow arrow

Miscellaneous



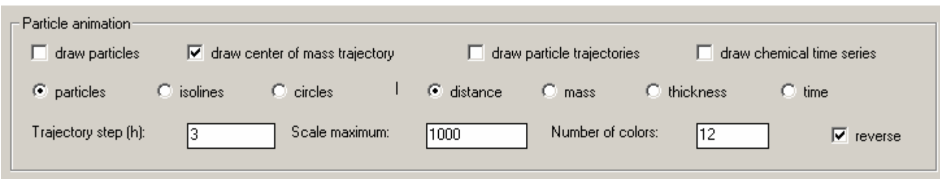
After run sustain animation window – if checked on, the animation window will not be closed after the running is finished

Draw land mask in OpenGL – fill land with specified color (color predefined, but can be changed in model graphics control file)

Draw vertically averaged velocities – calculate vertical averages of flow and draw them

Draw isolines with numerical values, every – draw isolines and attach numerical values to them indicating isoline value

Particle animation



Draw particles – to draw the particle animation, if active, check on this check-box

Draw centres of mass trajectory – calculate center of mass for oil or chemical spill and draw its trajectory

Draw particle trajectories – draw trajectories of individual particles

Draw chemical time series – draw time series of emulsified, evaporated, dissolved, decayed and on-bottom chemical amounts

Particles – draw slick with particles; particle color indicates distance, mass, thickness or time (see below)

Isolines – show slick mass distribution with isolines

Circles – show aggegates of mass with circles; circle diameter indicates mass; circle color indicates distance, mass, thickness or time (see below)

Distance – show distance as color

Mass – show mass as color

Thickness – show distance from surface as color

Time – show particle time in water column as color

Trajectory step (h) – timestep how often the trajectory will be updated in the animation

Scale maximum – distance (depth), mass, thickness, or time scale maximal value

Number of colours – number of different colours to be used in the particle animation (max 13)

10.3.3 Update and delete animation ts

It is possible to draw time series in the animation window from previous model run. This enables quick comparison between scenarios or view on the impact of parameter change. **Update** option updates the time series to the current time series, **delete** destroys previous one. The animation time steps and simulation times should be the same between model runs for the time series to appear properly.

10.4 TIMESERIES HANDLING AND OPTIONS

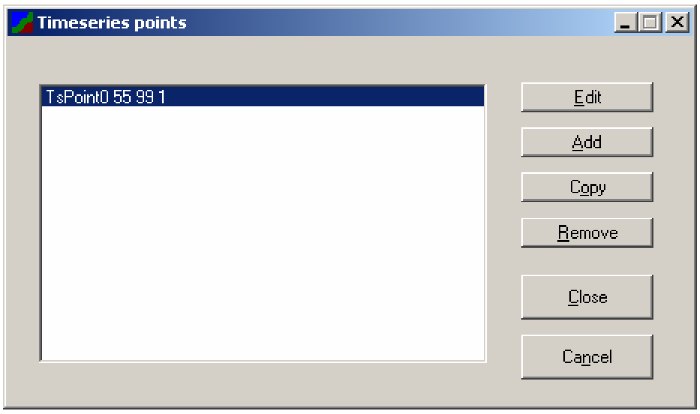
User has two different ways to add and edit timeseries points:

- Through menu item **Model – timeseries… - points**
- Through map interface

Both options have been described in the following sections.

10.4.1 Timeseries points handling through menu

Select **Model – Timeseries… - Points** to see **Timeseries points** window. In this window user is able to see the list of the existing TS points and user can add new TS points if needed. The window is presented below



List actions:

Edit - edit item data

Add - add new item

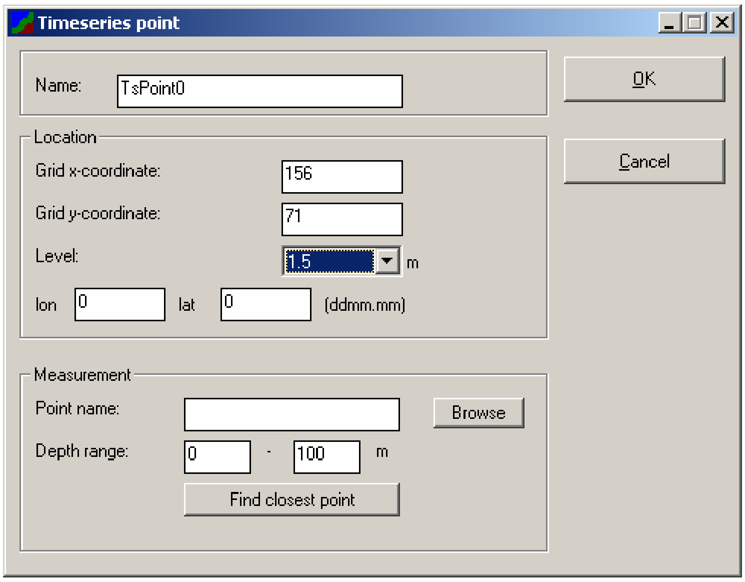
Copy - create a new item by copying an existing item

Remove - remove an item

Close - close the list window

Cancel - close the list window

The Timeseries point data window (see below) will be opened when either existing TS point is edited (**Edit**) or new TS point created (**Add**)



Name

Name – name of the TS point

Location

Grid x-coordinate – x-coordinate of the timeseries point

Grid y-coordinate – y-coordinate of the timeseries point

Level – level of the timeseries point. The TS point vertical location is in the middle of the vertical cell.

Lon / Lat – lon/lat coordinates of the TS point (optional)

Measurement

User can link the TS point to the related measurement data by using settings in this sub-category.

Point name – browse the measurement point by using Browse button

Depth range – defines the depth range of the measurement

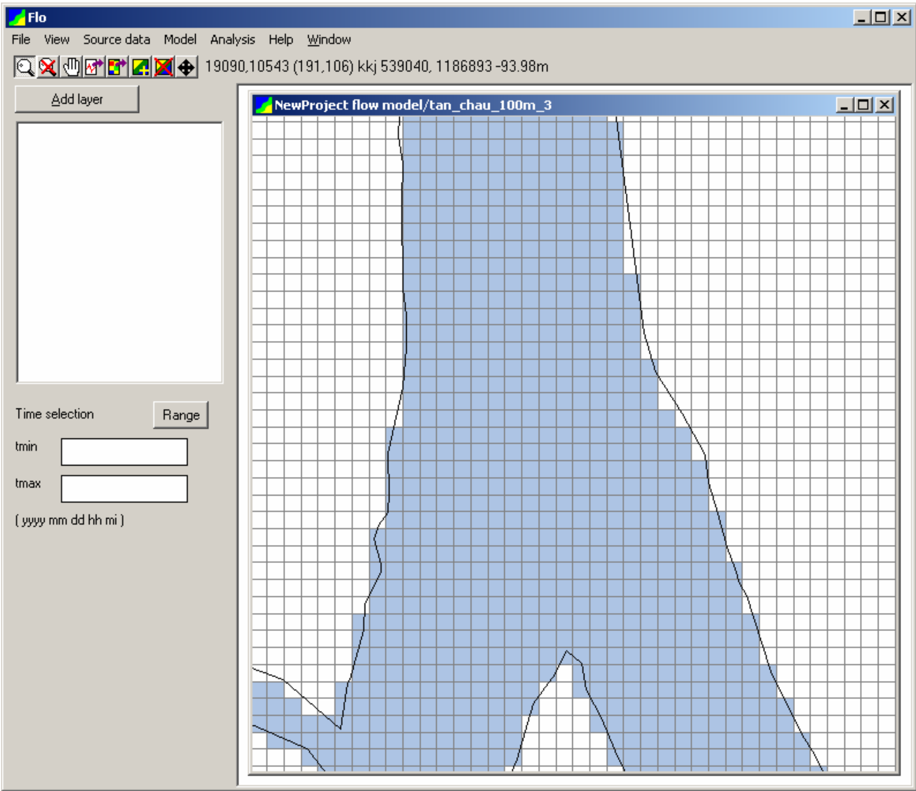
Find closest point – find automatically the nearest measurement

10.4.2 Adding and editing timeseries points in map

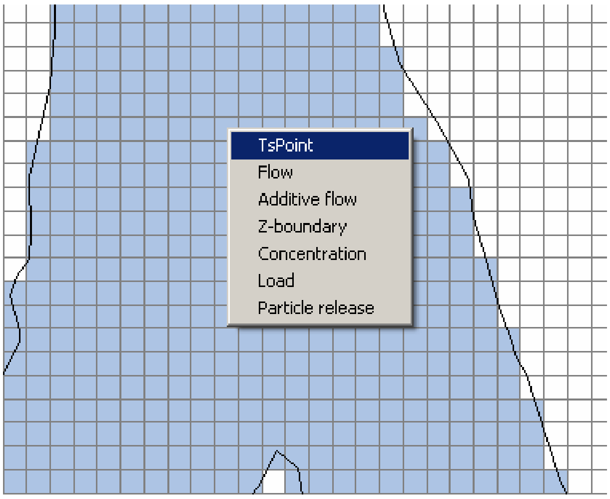
User can also add and edit timeseries options through the map based interface

Adding new timeseries point

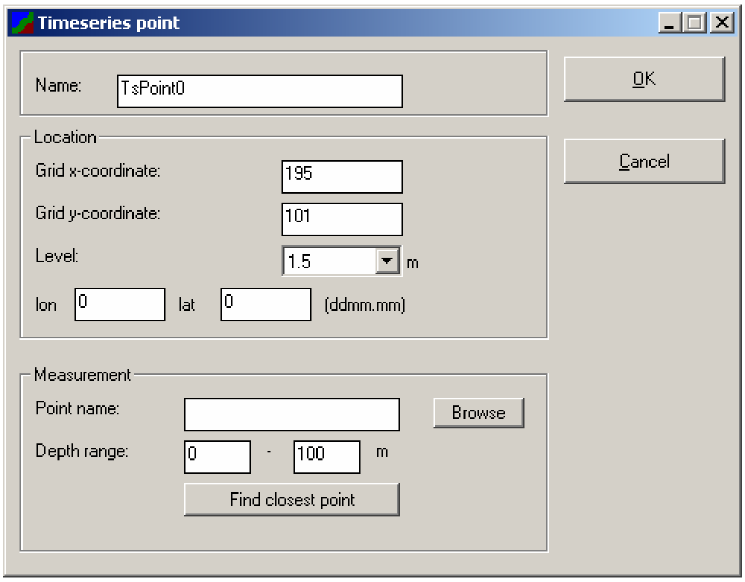
1. activate the main model window if not active already
2. Zoom into the area you want to add the new TS point



3. Select **Add item** tool  from the toolbar and click with left mouse button to the location where you want to add the new TS point. The following menu appears on the screen

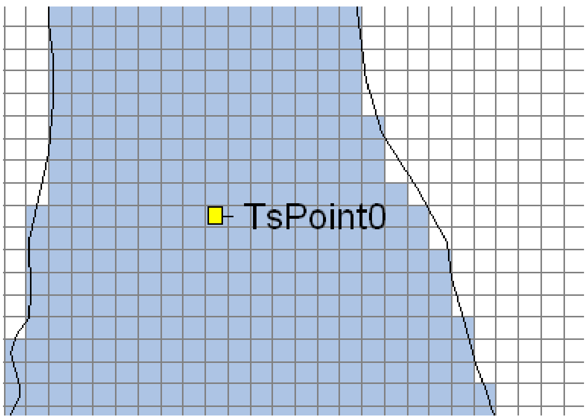


4. Select TsPoint with the left mouse button and window for timeseries point parameter set-up will appear.



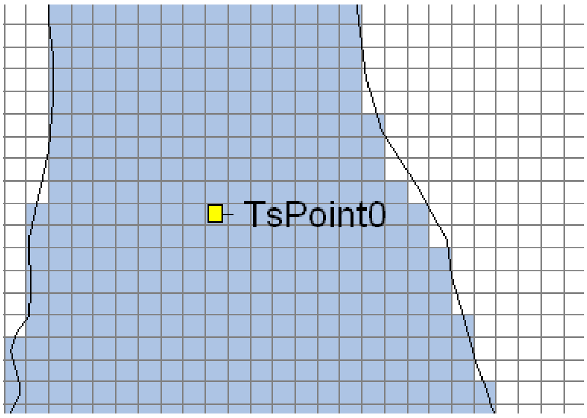
5. See the previous (Section 10.4.1 – Timeseries points handling through menu) sections for more detailed information for each option in the menu. When the parameters have been set up, press **OK** to finish adding a new TS point. To cancel, press **Cancel**.

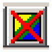
6. The TS point appears to the map

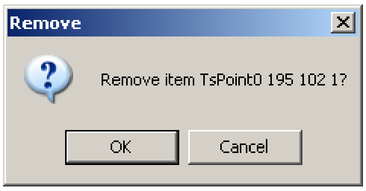


Removing timeseries point

1. Zoom into the TsPoint you want to remove



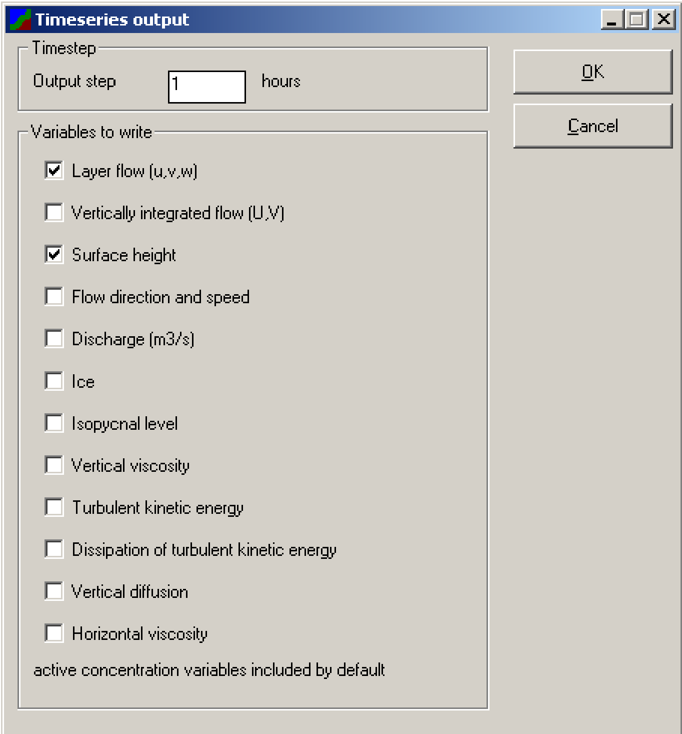
2. Select the Remove item tool  from the toolbar
3. Click on the Timeseries point you want to remove with the left mouse button and following window appears



4. Press **OK** to remove the item and **Cancel** to cancel the action

10.4.3 Timeseries options

In **Timeseries output** window (**Model – Timeseries… - Output**) user can select the timestep for TS output and the recorded variables. The window is presented below:



Timestep

Here user can define the frequency (**Output step**) how often the simulation results will be recorded into the timeseries file. Depending on the application the recommended timestep is 1-24 hour(s). If the running time is very short and/or user wants to see the simulation the shorter **Output step** should be used.

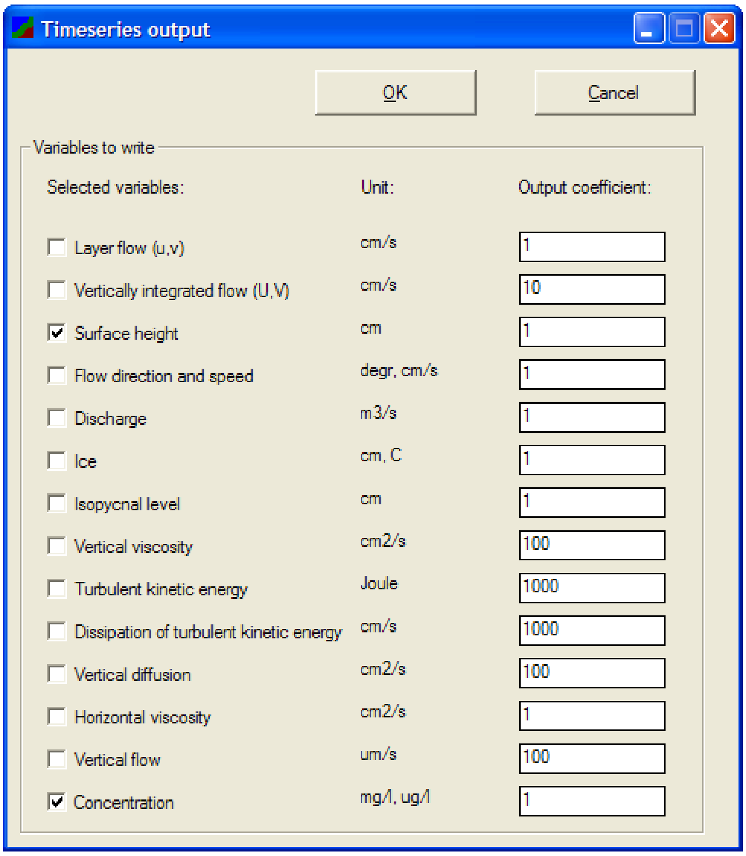
Variables to write

User can define which variables will be written into the timeseries file by checking the wanted variables.

Active concentration variable are included into the timeseries file by default.

10.5 OUT-FILE

Out-file (*.out) is the file where all parameter values and selected model results written to. User can select which variables which will be written to the out-file numerical timeseries:



Output coefficient multiplies the variable values in order to provide convenient output unit.

11 FETCH AND DYNAMIC FIELDS

This chapter provides information for how to save fetch and dynamic output information, and then how to read and use the saved data.

The chapter is divided to three parts:

11.1 DESCRIPTION OF FETCH AND DYNAMIC OUTPUT

With the stored fetch and dynamic output the model can be run faster to do different scenarios for the similar wind and flow conditions. Descriptions for fetch and dynamic output are given below:

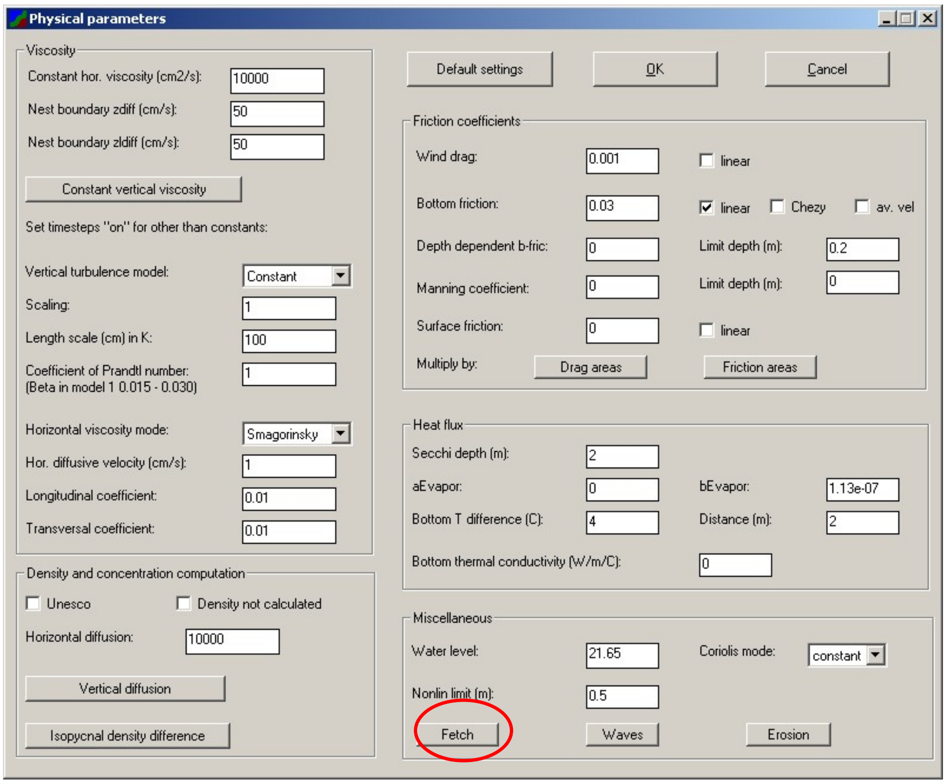
Fetch: Wind fetch is the distance from the shore line to the point of observation over which the wind blows with constant speed and direction. Writing the fetch means here that the wind field over the model area is calculated and saved to the file for each wind occasion. This makes the oncoming simulations much faster.

Dynamic output: Dynamic output contains the flow field calculated every decided time step. Writing the flow field again makes the oncoming simulations faster.

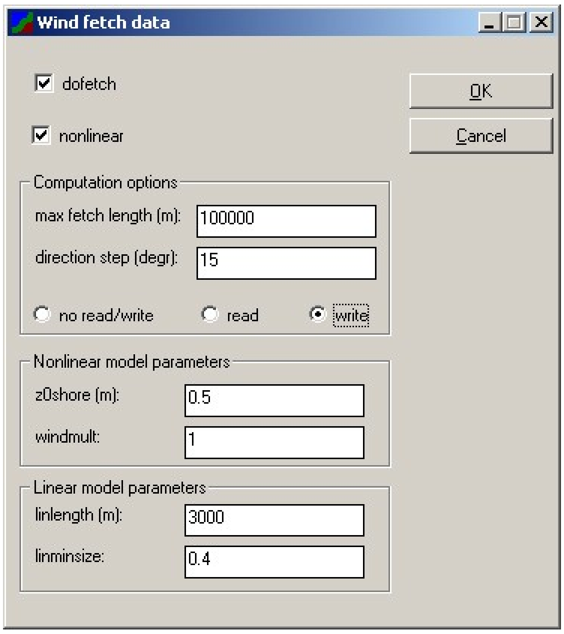
11.2 SAVE FETCH AND DYNAMIC FIELDS

Steps to save fetch and dynamic fields are provided below:

1. Write fetch
   a. Select **Fetch box** from **Model – Physical parameters…** window



b. Select the **write** box and **dofetch** as shown below



c. Click **ok**

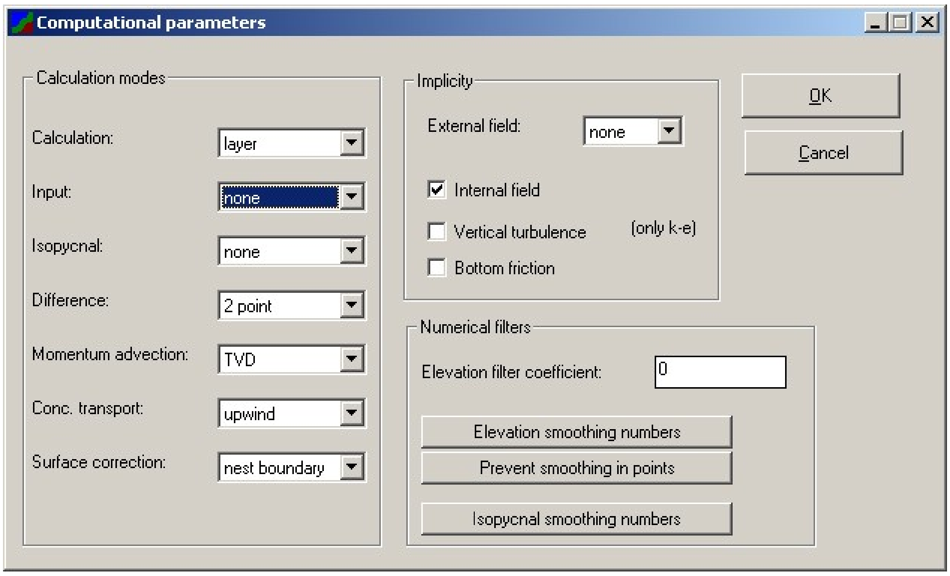
d. Erosion and Waves in physical parameters window:

   i. while you are writing the fetch for the oncoming model work you don't necessarily need to run wave and erosion models at the same time. You can do that later on if necessary.

2. Computational parameters

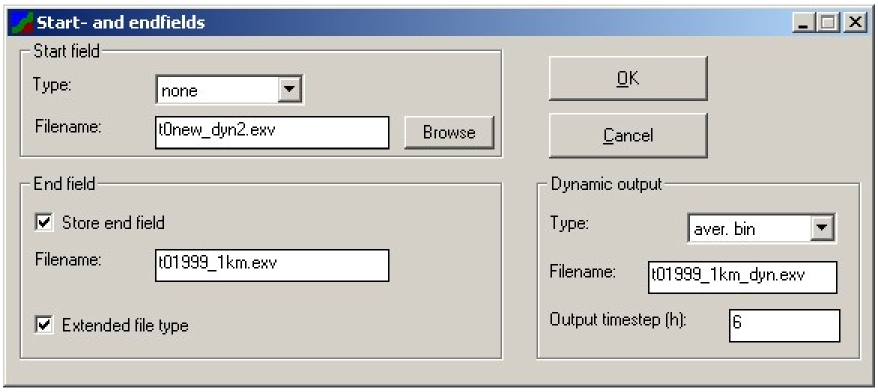
   a. Select from menu *Model – computational parameters…*

   b. Select **None** for Input dropdown menu



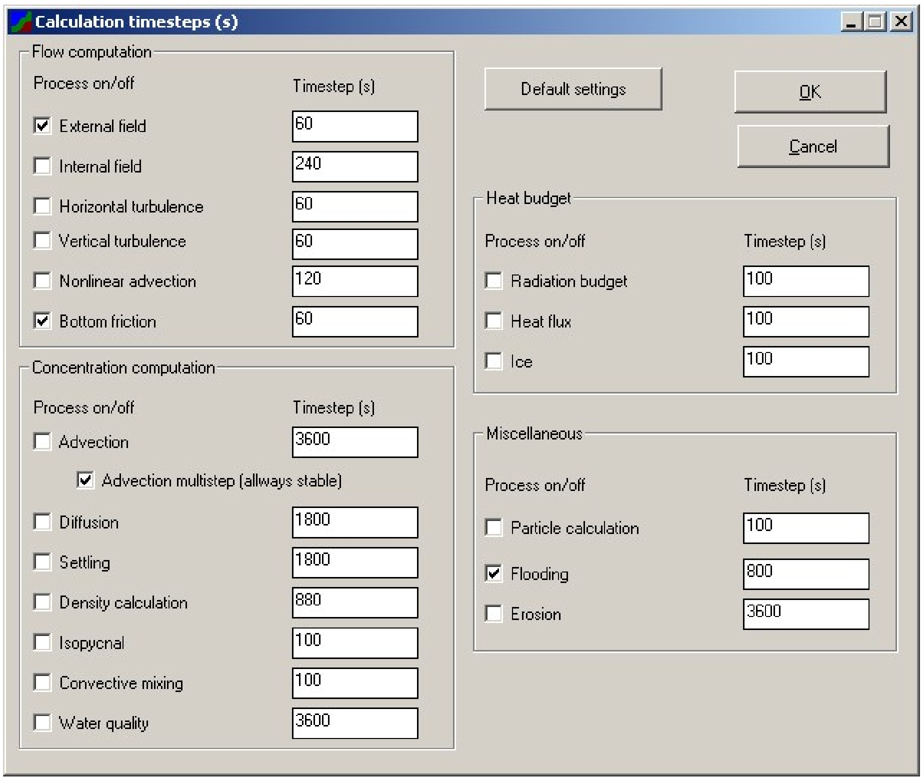
b. Click **OK**

3. Dynamic output
   a. Select from menu Model – Start, end and dynamic field
   b. Select for Dynamic output Type aver.bin from dropdown menu
   c. write the file name and select Output timestep (h)



   d. Click **OK** to accept the changes

4. Other settings
   a. Time steps: the model need to be stable to write the fetch and dynamic field (here example for the lake Tonle Sap Lake model while running with 1 km grid)



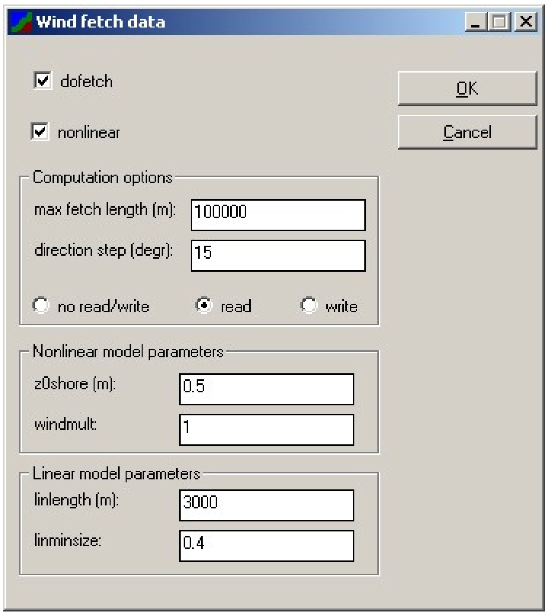
b. Calculation time: set the time you need to use the model in the future for the calculation time

c. Statistics: make sure that you have the needed statistic selected which reasonable time step

11.3 Read fetch and dynamic fields

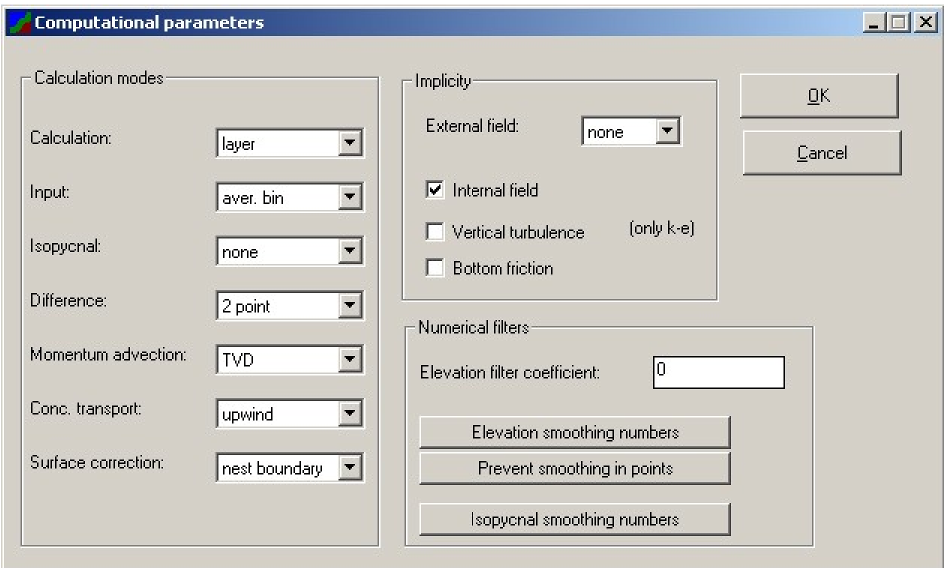
The steps to read the fetch and dynamic fields are provided below:

1. Reading fetch
   a. Select Fetch box from Model – Physical parameters… window
   b. Select Read and Dofetch boxes as shown below



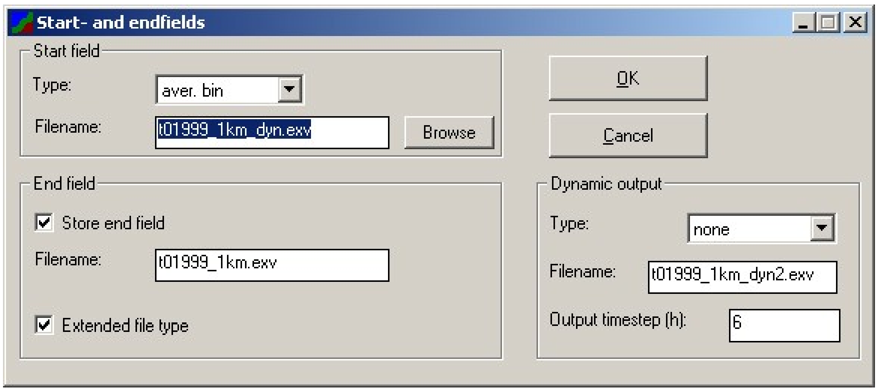
c. Click **OK**

2. Computational parameters
   a. Select from menu Model – computational parameters…
   b. Select aver.bin for Input dropdown menu



   c. Click **OK**

3. Dynamic output
   a. Select from menu Model – Start, end and dynamic field
   b. Select for Dynamic output Type none from dropdown menu
   c. write the file name of the saved dynamic output in the A part (t01999_1km_dyn.exv) to Start field’s file name and select aver.bin for the dropdown menu



d. It is also in this point possible to write again the Dynamic output if there is need to reduce or increase the output timestep. In that case, the new file name would be written to Filename field and new time step to Output timestep (h) field. Remember to select aver.bin for the Type dropdown menu.

e. Click Ok to accept the changes

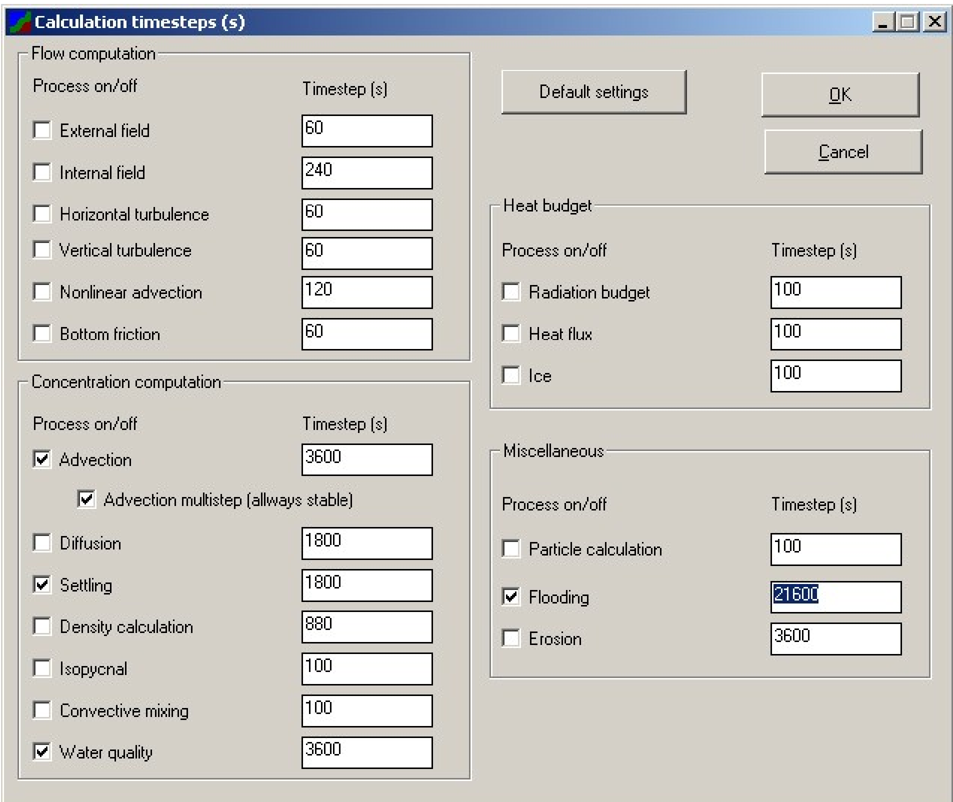
4. Time steps

a. Select from menu Model – Time steps…

b. Once the dynamic output has been saved the model need not to calculate again the flows so all the flow processes can be turned off

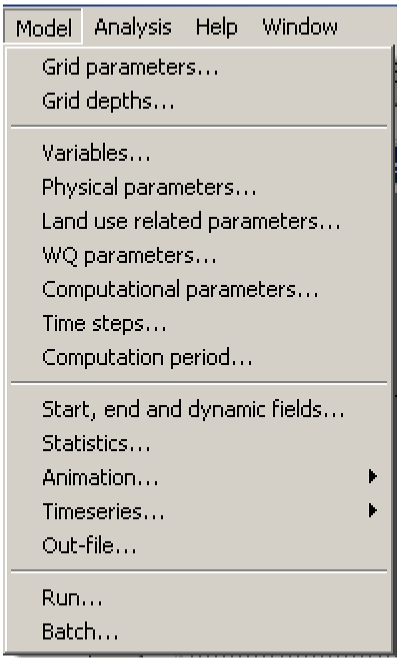
c. The flooding time step can be put to the same than the Dynamic output time step (here 6 hours -> 21600 to the flooding time step)

d. Other time step settings depends the calculated features



5. Other settings
    a. Calculation time: set the time you need to use the model in the future for the calculation time
    b. Statistics: make sure that you have the needed statistic selected which reasonable time step

12 RUNNING THE MODEL



This chapter describes how the model computation can be started and model computation time set up.

The chapter is divided to three parts:

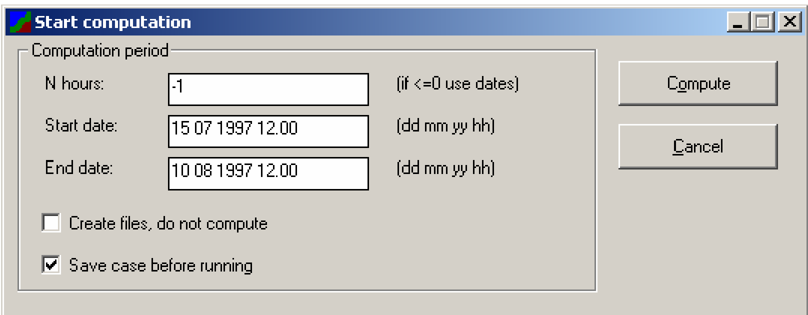
12.1 Run the model

12.2 Batch

12.3 Operating in the model computing window

12.1 Run the model

The model can be run by selecting the **Model – Run…** from the main menu. The “Start computation” window appears:



The computation time can also be set up in “Computation time” window under **Model – Computation period…**

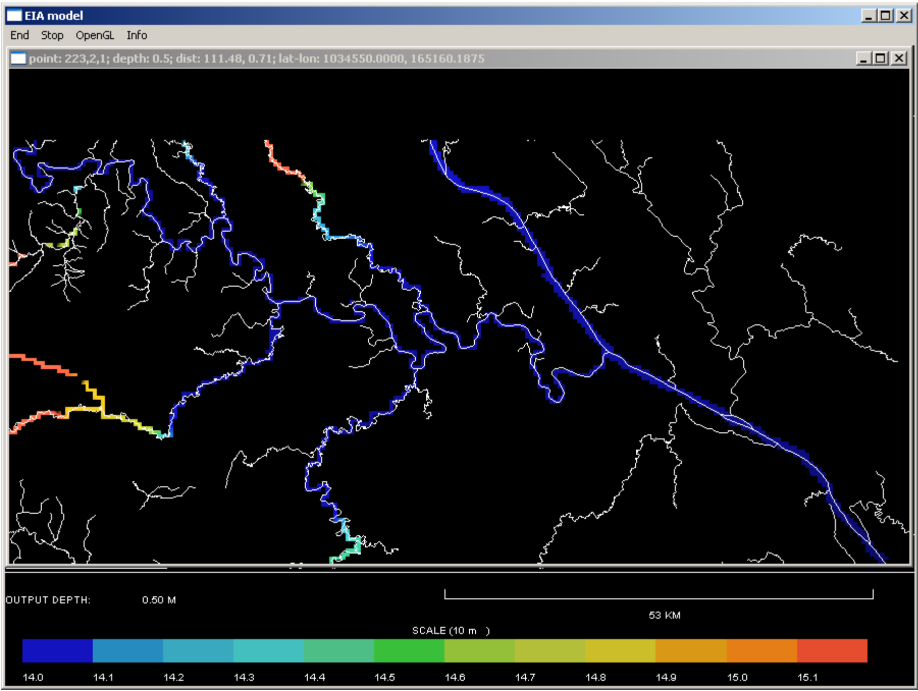
Under the computation period section of the “Start computation” window the following parameters can be set up

- N hours - number of hours to compute, if less than zero then use dates below
- Start date - computation start time in format 'dd mm yyyy hh'
- End date - computation end time in format 'dd mm yyyy hh'
- Create files, do not compute – checked if only computation files are needed

- Save case before running – checked if wanted that the model application will be saved before running

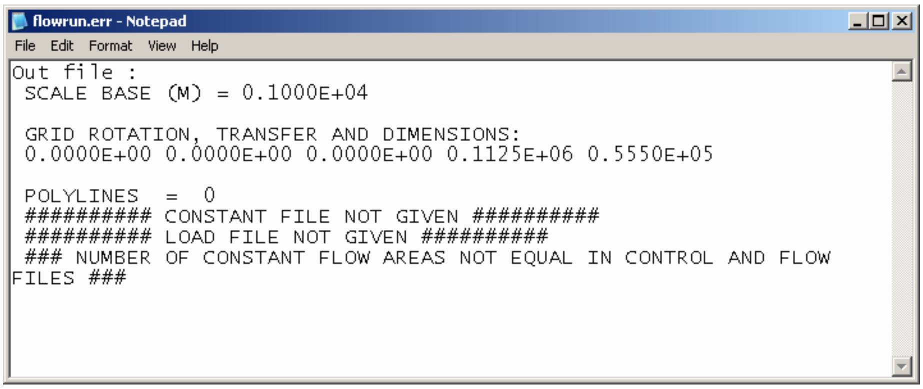
When the parameters are set up, the model computation can be started by clicking **Compute**. By clicking **Cancel** will cancel the settings and get the user back to the main model window.

The following window will appear to show the model progress. The operation options in the model computing window are described in Section 12.3 – Operating in the model computing window.



If there is some trouble in the model running the following window appears

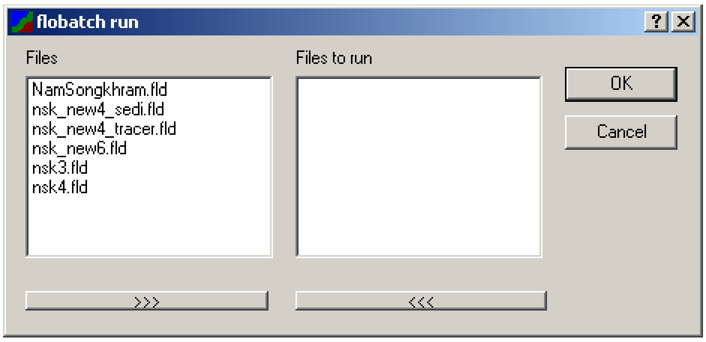
Clicking OK, the "flowrun.err" window appears and user can see the error file of the model run. More about the trouble shooting can be found from Chapter 18 – Troubleshooting. Below one example of the error file is shown:



The most common trouble shootings, when running the model, are described in Chapter 18 “Troubleshooting”.

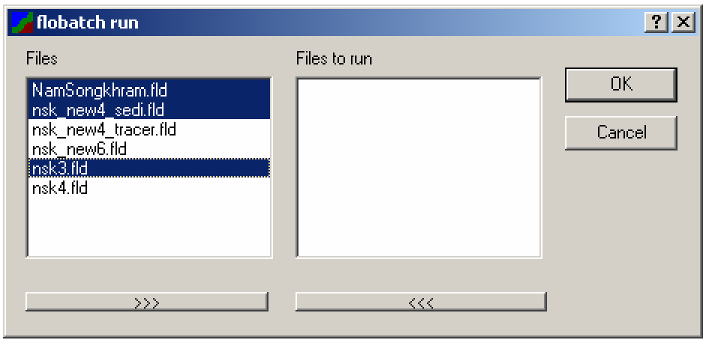
12.2 Batch

This function allows you to run several model simulations automatically one after each other. The user can select the models to run by selecting **Model – Batch…** and the following window appears

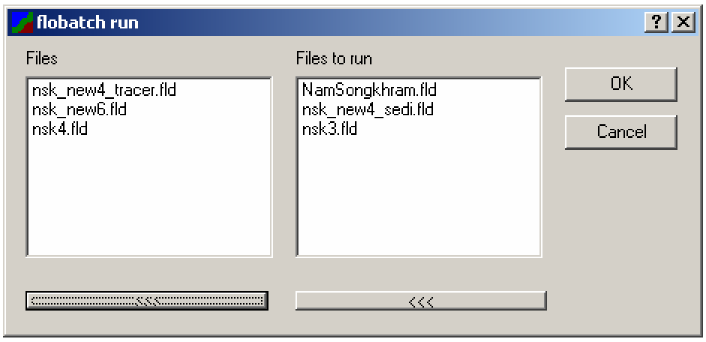


The models need to be in the same folder to be able to batch them. Follow the steps below to batch the models:

1. Select first the files you want to batch keeping ctrl pressed down or move them there separately repeating steps 1 and 2



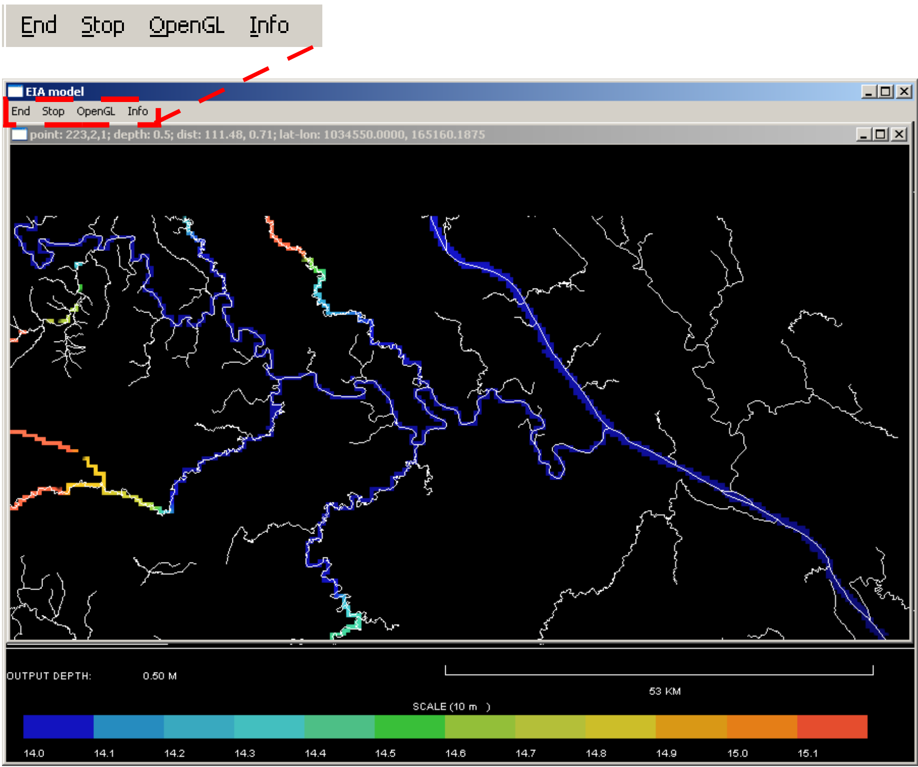
2. press **>>>** button to move the files to “Files to run” section of the window



3. To move file(s) back to “Files” section of the window, select the file(s) to be moved and press **<<<** button
4. Press **OK** to run the batched files

12.3 Operating in the model computing window

The user is able to zoom in, zoom out, and see the model simulation in different vertical layers in model computing window while the model is computing.



End:

- Delete window – terminates the computing process and closes the computing window
- Save window – stop computing but leaves the image on the screen

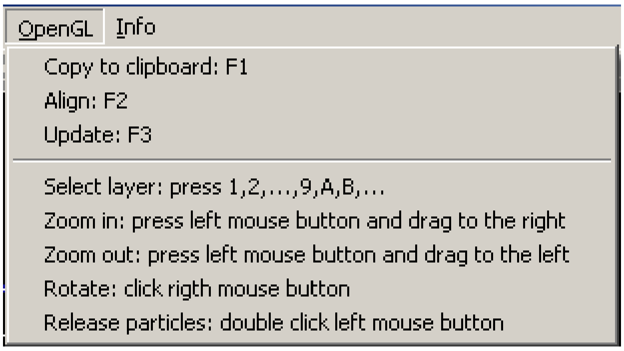
Stop:

- Halt – will pause the computing

- Resume – continue the computing if it was halted

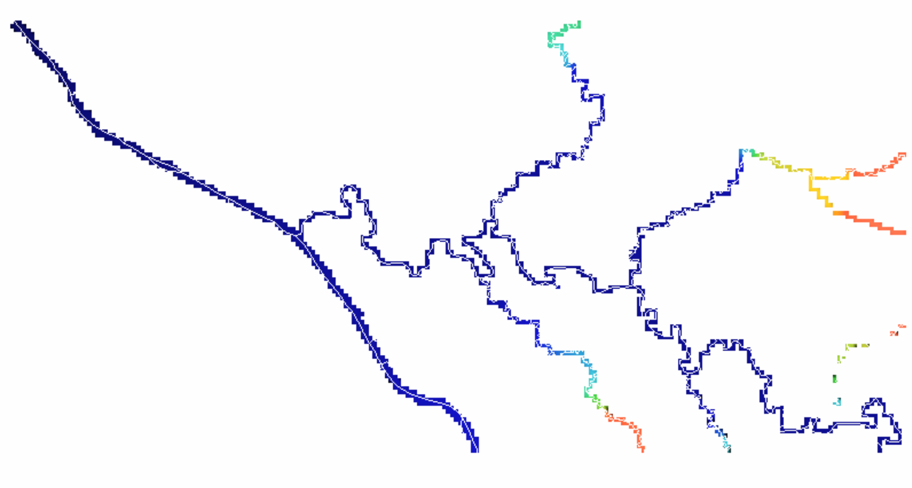
OpenGL:

Under OpenGL menu the user can find instructions how to manage the OpenGL window as shown below.



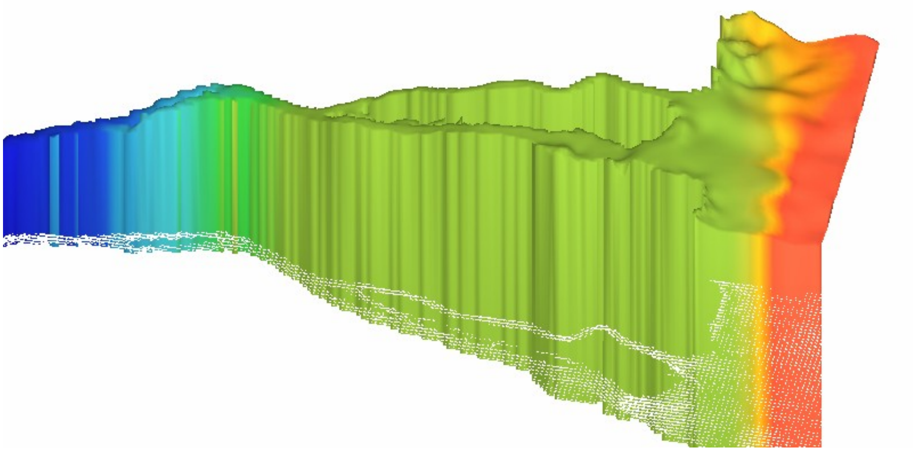
The first three items in the menu list work also when clicking them. After those items the menu offers only instructions.

- Copy to clipboard: F1 – copy the window capture to the clipboard



- Align F2 – align the view again to horizontally plane if user has used the 3D rotation
- Update F3 – update the view for the current time. In animation options (see Section 10.3.1 Animation options) user can select the animation frame step (h) which defines how often the animation frame is updated.
- Select layers – user is able to see the simulation in different vertical layers. Use numbers from 1-9 to layers 1-9, alphabets a-z for following layers starting from 10-. Number 0 is used for bottom layer, i.e. it shows the bottom layer simulation no matter in which layer the bottom is
- Zoom in – user is able to zoom in to the simulation window by pressing left mouse button and dragging from left to right
- Zoom out – press left mouse button and drag from right to left
- Rotate – user can rotate the 3D view. User is able to define other parameter for the elevation and other for the colour (see Section 10.3.2 – Advanced

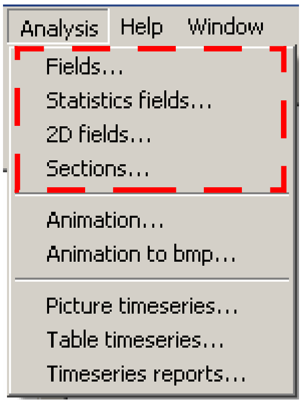
animation options). Below example where the elevation shows the water level and colour salinity concentration.



- Release particles – double click left mouse button in the location where particles needed. The following options should be checked on that this work
    - time step for particle calculation
    - particle animation checked on in advanced animation options (see Section 10.3.2 – Advanced animation options)

Info: information of the model engine and contact information of the model developers.

13 MODEL OUTPUT ANALYSIS TOOLS



13.1 To get started

Field analysis draws 2D picture of computation end state concentrations. It enables also drawing of model grid and pre-computed flow fields.

With the field tool it is possible to present the results in different form and do calculation. It is possible to use different masks to e.g. calculate the net sedimentation in each landuse class or elevation zone (only available in 2D fields).

The basic setup of different windows in the field analysis tool is presented in figure 15. To start the analysis tool, there are two options:

- Select the **Analysis** and **Fields, Statistics fields** or **2D fields** option
- Start the analysis tool by open the wanted *.exv file (e.g. ..\Tonle\analysis\1998_2d.exv) and the “Field draw and Compute” window will appear:

These options open the **“Field draw and compute”**-window**.** The three types of model output field files are:

- Fields – computation end fields; these are defined as presented in Chapter 10.1.
- Statistic fields – user needs to select the option to compute the **3D statistics fields** from the “Statistic parameters” window (see Chapter 10.2).
- 2D fields – user needs to check on the **Output 2D fields** from the “Statistic parameters” window (see Chapter 10.2).

These files are allways Ascii (text)-files in contrast to dynamic fields that can be either binary or Ascii.

The basic use of the analysis tool is described in Section 13.2 – Draw and compute. In addition to the basic analyses the user is able to do more advanced analysis by using **Set compute** option which allows the user to use following tools:

- Mask – by using mask the model area can be divided to sub-areas based on landuse, elevation or other parameter of the model
- Region – the model area can be divided to regions based on coordinates
- Formula – user is able to do calculations e.g. between different variables using the formula tool.

This chapter is divided to the four parts:

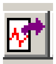
- Draw and compute – basics of the draw and compute options applicable in each field analysis option

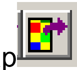
- Different fields – describes the differences between the field analysis available in EIA 3D model system
- Set compute – advanced options for analysis under 2D fields analysis tool
- Sections – describes the section analysis tool

13.2 Draw and compute

Basic steps to draw and compute analysis are:

1. Select variable from the **Variable** drop-down menu
2. Set up the Options as scale, length scale, draw options etc.
3. Add header and background map if needed
4. Press **Draw & Compute** button
5. The results appear on separate graphics (and numerical results) windows
6. The result can be copied as a bitmap or metafile to the clipboard and then attach to word document by using the following tools in the toolbar:

   a. Copy as metafile 

   b. Copy as bitmap 

Best settings for draw option are different for each application and thus, the best result can be achieved by trying different values for the options you want to use.

The **"Field draw and compute"** window is presented below with the default settings for each parameter.

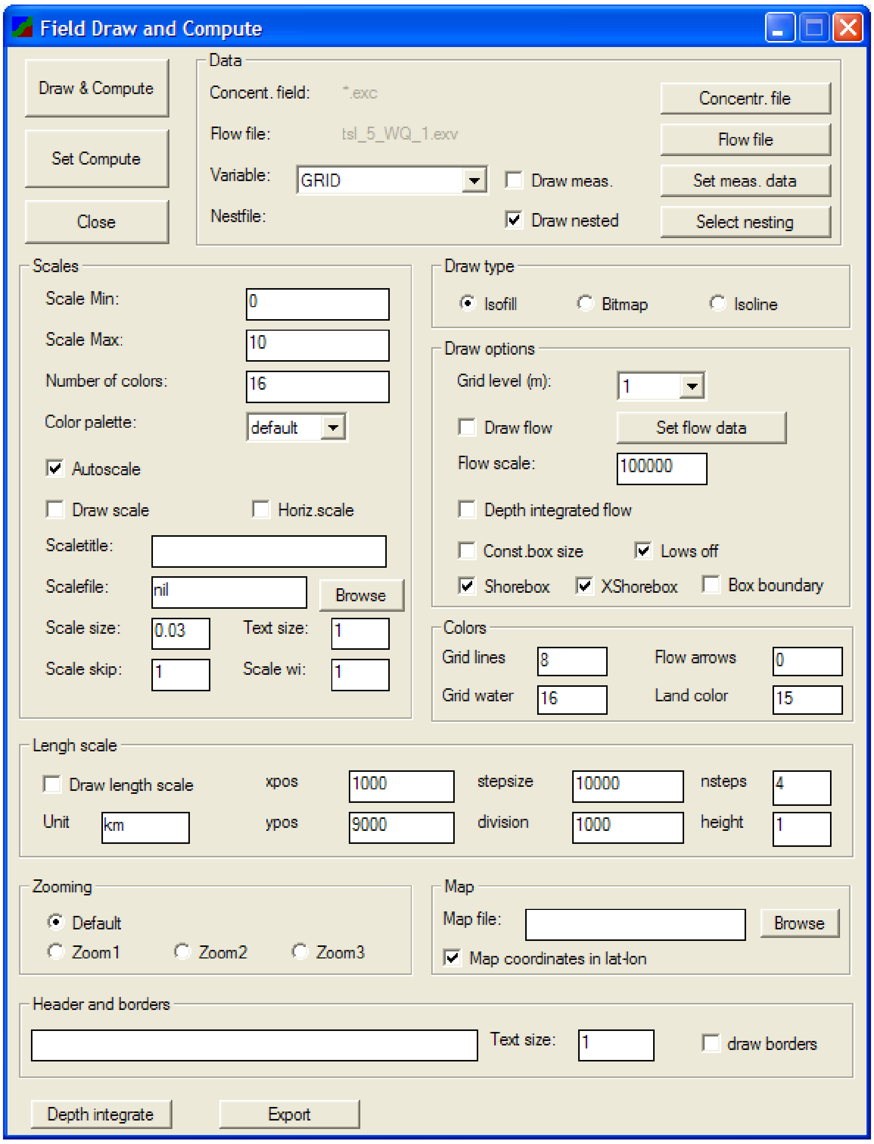
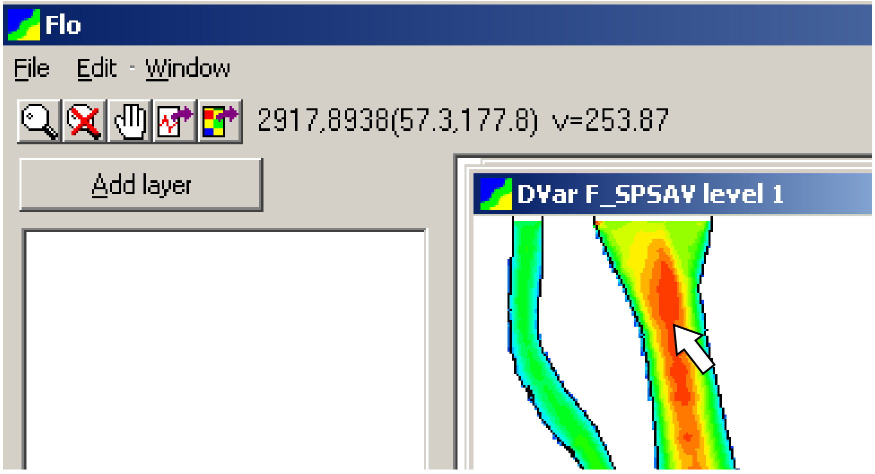


Figure 15. The basic setup of different windows in the field analysis tool.

The variable value and location in each point of the analysis drawing can be seen in next to the toolbar as illustrated below:



13.2.1 Data

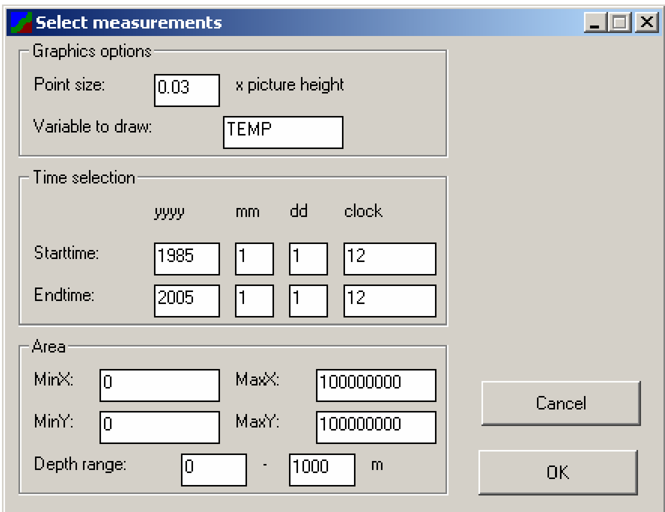
Concent. field – shows the used concentration file (*.exc) if used. Press **Concentr. file** button to load another concentration file. This is special water quality model output file.

Flow file – shows the name of the used flow file (*.exv) if used. Press **Flow file** button to load another flow file.

Variable – user needs to select the variable she/he wants to draw and analyse. The list of the parameters with explanations is presented in Chapter 27 – Parameters in EIA 3D model.

Nestfile – name of the nesting information in case of nested model

Set meas. data/ Draw meas. – if checked, draw measurements of the selected variable as circles inside which the measurement value is indicated with numbers. Set the options by clicking **Set meas. data** button, which opens the following dialog window.



User can specify circle size as fraction of total figure height, measurement variable name and time, topographic and depth range of measurements to be shown of total measurements.

Draw nested – Draw as nested or each nested grid separately. Set nesting definitions file by clicking **Select nesting** button.

13.2.2 Scales

Drawing scale option:

Scale min – scale minimum either from model output (autoscale checked) or given by user (autoscale not checked)

Scale max – scale maximum from model output or given by user

Number of colours – defines the number of colours used in the figure

Colour palette – user can select the colour palette used:

- Default – normal colour palette from blue to red giving blue to smaller values
- Inverted – inverted colour palette (from red to blue) giving red for smaller values; use e.g. for dissolved oxygen
- Jet – more vivid colours
- Inv.jet – inverse jet scale
- Hsv – hue, saturation, value scale
- Grey – grey scale from white to black
- Inv.grey – grey scale from black to white

Autoscale – if checked on (Default), the minimum and maximum values are the min and max values in the results. If not checked, the user can define the min and max values for the illustration.

Draw scale – draws the scale

Horizontal scale – if checked, scale will appear on bottom of the figure in horisontal direction, if not checked the scale appears on right from the figure in vertical direction

Scaletitle – title for the scale, appears on top of the scale bar

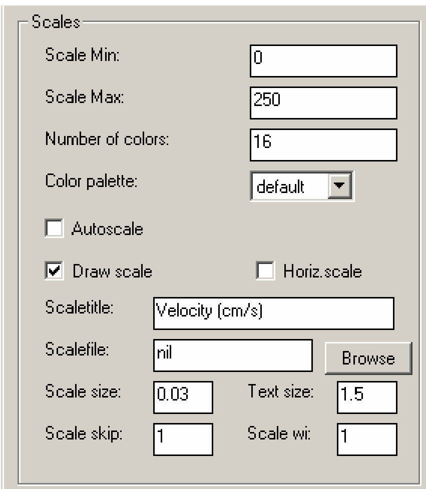
Scalefile – instead of given palette selections user defined scale palette gan be given in a file

Scale size – scale bar size (impacts also on length scale)

Scale skip – the number of scale classes to skip in labelling

Text size – defines text size in labelling and titles

Scale wi – scale bar width



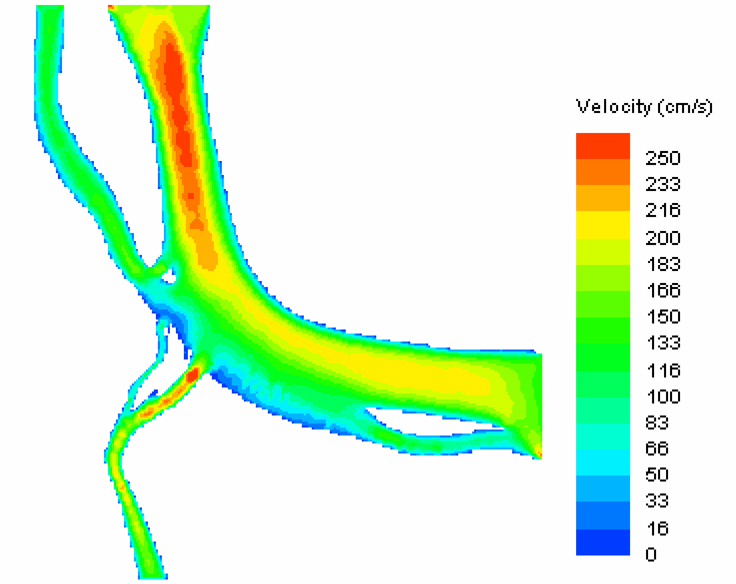
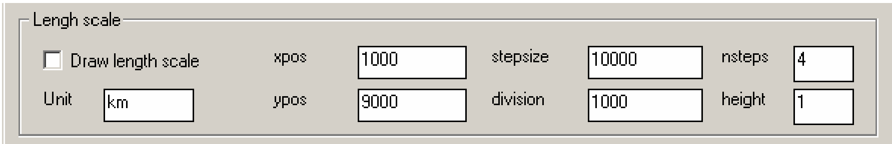


Figure 16. Example result of the field draw with the options described above.

13.2.3 Length scale



Draw length scale – if checked, the length scale will be drawn on the bottom left corner of the figure

unit – km if the division is 1000 as above, m if the division is 1

xpos – x-coordinate location of the length bar in meters (in grid coordinate system) from the origo

ypos – y-coordinate location of the length bar in meters (in grid coordinate system) from the origo

stepsize – length of the one step in the scale bar in meters

division – defines the unit of the length scale (1000 for kilometre, 1 for meter, etc)

nsteps – number of the steps in length scale

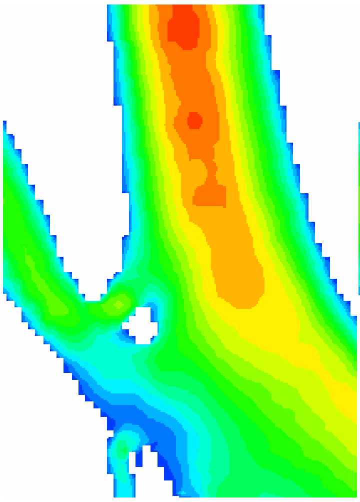
height – height of the length scale bar

13.2.4 Draw type

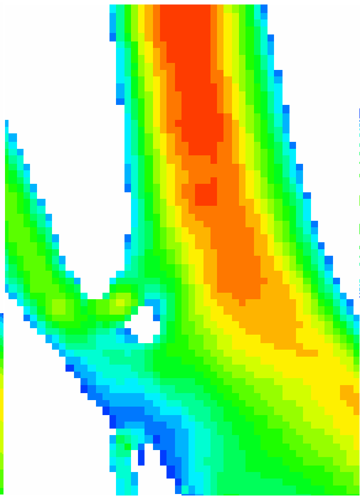
There are three different draw types to select:

- Isofill – vector based filled draw type
- Bitmap – raster based draw type
- Isoline – vector based non-filled draw type

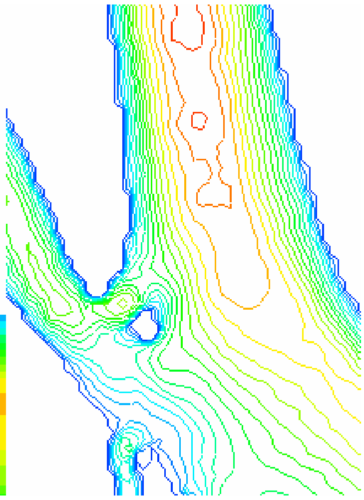
Each of the draw type is presented below:



Isofill:

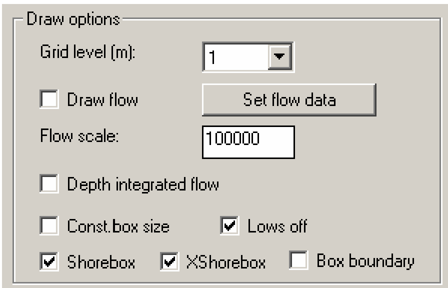


Bitmap:



Isoline:

13.2.5 Draw options



In draw options the user is able to

- define the grid level for the analysis (in Fields and Statistic fields only),
- define whether the flow arrows will be drawn or not,
- select Depth integrated flow to be used and
- definitions for the illustration box

More detailed description for each of the options follows:

Grid level – in field and statistic field analysis user can define the grid level from where the results will be drawn. In 2D field analysis user can select only surface, middle, and bottom layers, those are eligible already in the variable drop-down menu.

Draw flow – the flow arrows will be drawn if checked.

Flow scale: scale for the flow arrows, best scale depends on the flow velocities and the size of the applications.

Depth integrated flow – the results will be drawn as depth integrated flow. The layers to be included can be defined by clicking the Integrated flow button on the bottom of the “Field draw and compute” window.

Const. box size – leave to default value

Lows off – leave to default value

Shorebox – leave to default value

XShorebox – leave to default value

Box boundary - leave to default value

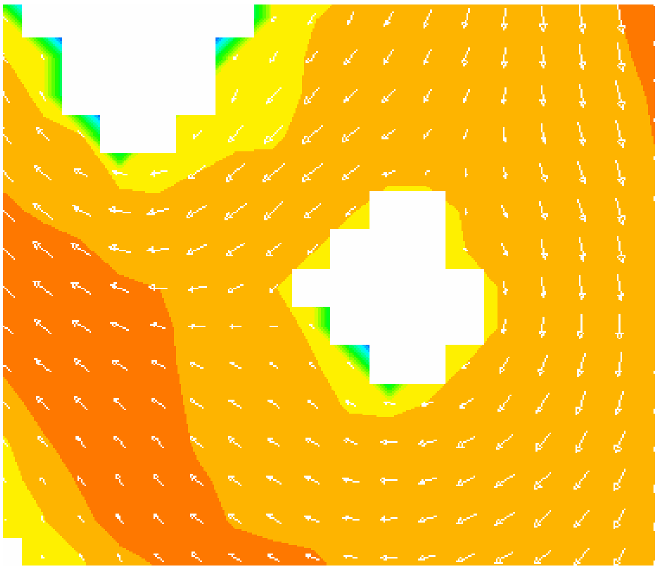
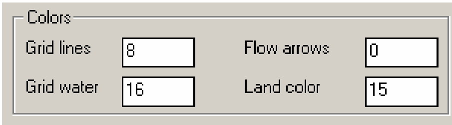
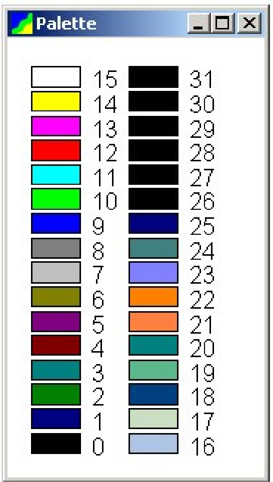


Figure 17. Flow field for the Chaktomuk model application in Cambodia.

13.2.6 Colours



In Colors section user is able to change the colour of the grid and flow features. The numbers correspond to the following colours:

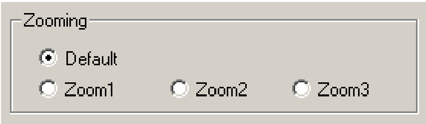


0 - black
1 - dark blue
2 - dark green
3 - dark green
4 - dark brown
5 - dark violet
6 - olive
7 - light gray
8 - dark gray
9 - blue
10 - light green
11 - cyan
12 - red
13 - purple
14 - yellow
15 - white
16 - light blue
17 - gray
18 - dark blue
19 - light green
20 - dark green
21 - orange
22 - orange
23 - gray blue
24 - dark green
25 - dark blue
26 –31 black

User is able to select the colours for the following features:

- Grid lines
- Grid water
- Flow arrows
- Land colour

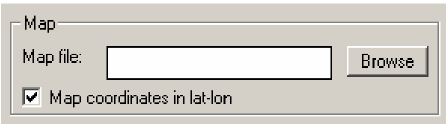
13.2.7 Zooming



Analysis area can be redistricted to certain zoom area:

- Default: the whole model application area
- Zoom1 – Zoom3: user can select the zoom area to zoom. See Section 6.1 – Zoom for more information of the Zoom areas and how to set them up

13.2.8 Map



User is able to use background map in the analysis drawing. The map should be *.dig file (see more about that file type in **RLGis manual**). The map can be browsed by clicking the Browse button. If the map is in lat-lon system, check the "**Map coordinates in lat-lon**" check-box. If UTM (x, y) coordinate system is used, uncheck the box.

Example of using the map file in the field analysis drawing is presented in Figure 18.

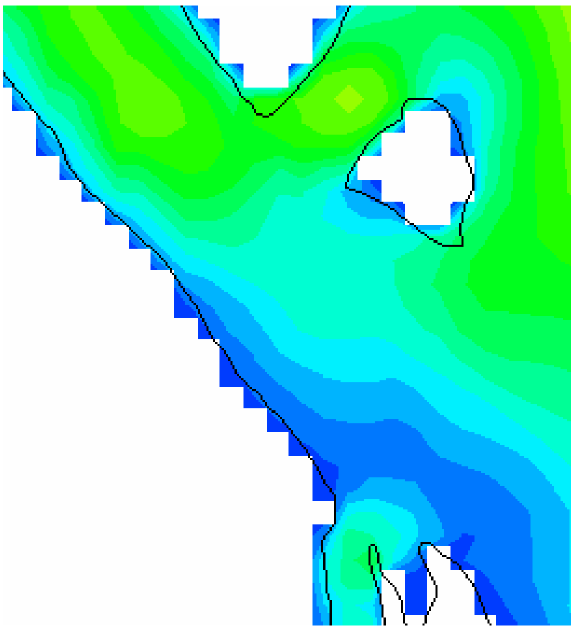
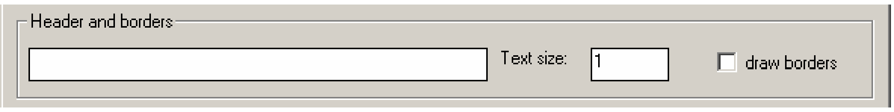
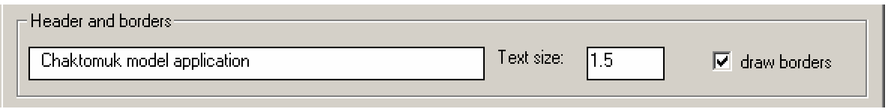


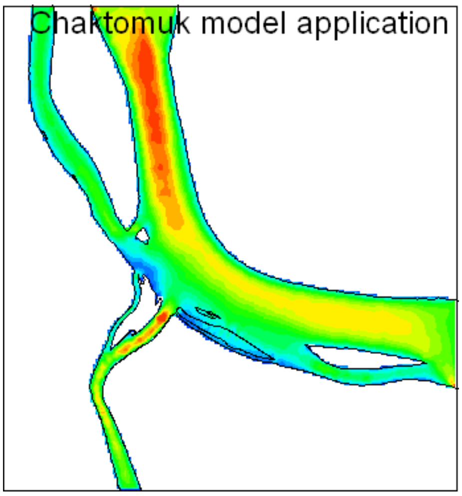
Figure 18. Example of using the background map in the field drawing.

13.2.9 Header and borders

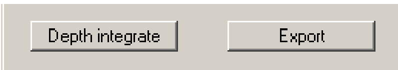


In this section the user can define a header for the analysis drawing by typing it to the empty text field. The text size can be defined and borders added around the drawing. Example of using the header and borders is given below.

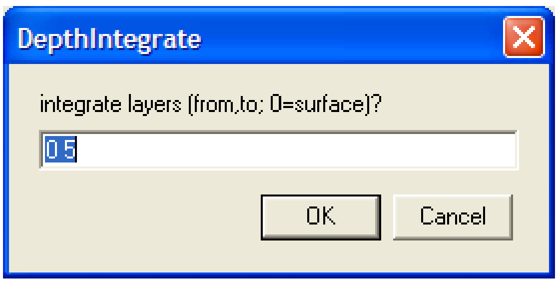




13.2.10 Depth integrate and Export



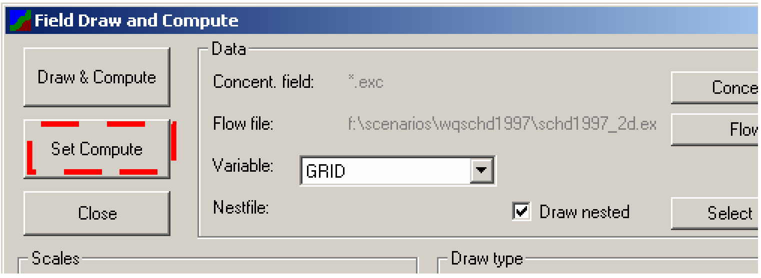
Depth integrate – define layers that will be integrated/ averaged for output



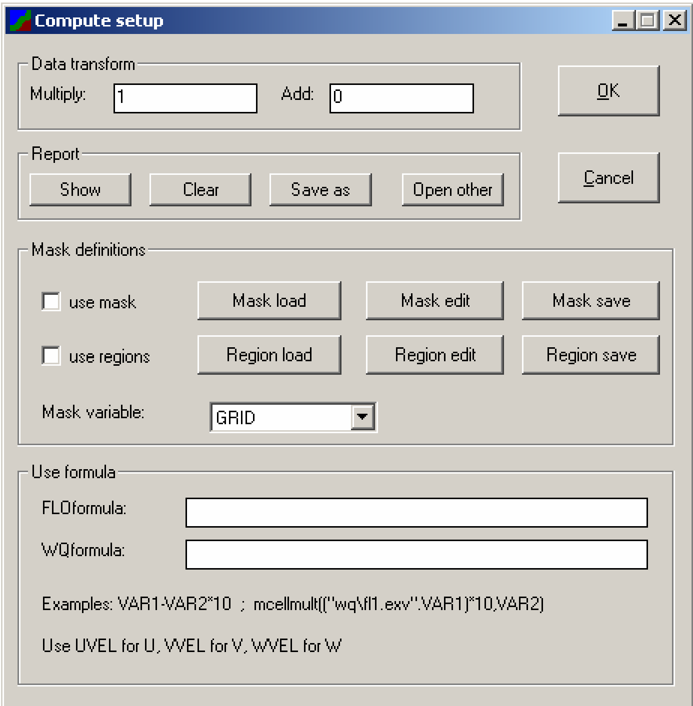
Export – not functional

13.2.11 Set compute

With “Set compute” tool the user is able to use masks and formulas to do more advanced analysis for the data. Press **Set Compute** button in the “Field draw and Compute” window to open the “Compute setup” window.



The following window will appear



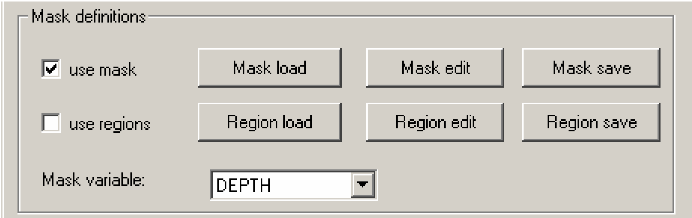
Under Set compute tool there are three main functions as described in more details in the following sections:

- Using the masks in “Set compute”
- Using the region in “Set compute”
- Using the formula in “Set compute”

13.2.12 Using the masks in “Set compute”

By using mask the model area can be divided to sub-areas based on landuse, elevation or other parameter of the model. This can be used for looking the specific model results for e.g. each of selected elevation zones or landuse classes separately. To use the mask follow the instructions below:

1. Activate the mask by ticking the use mask box



2. Load the mask by clicking the **Mask load** button and select the mask file you want to use (*.msk)

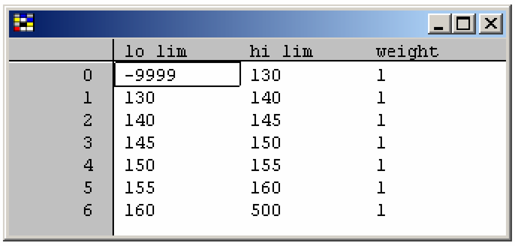
If you don't have any *.msk file available, you can create new file following the steps below:

- Create an empty txt-file with e.g. notepad
- First line defines the matrix's dimensions (n*3), following lines includes the data as shown below where example of 7*3 matrix of using elevation as mask

  matrix(7 3
  -9999 130 1
  130 140 1
  140 145 1
  145 150 1
  150 155 1
  155 160 1
  160 500 1)

- Save the file as *.msk (e.g. elev.msk)
- Load the file as defined in step 2

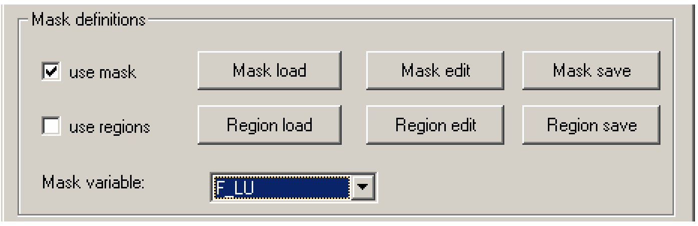
3. To see the mask file click the **Mask edit** button and the following window appears



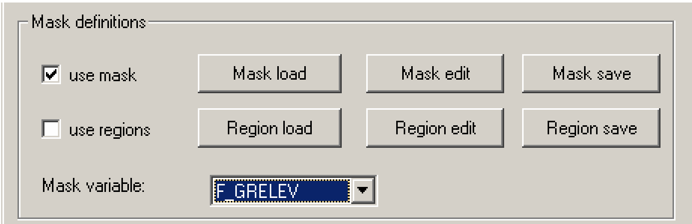
   a. You can add rows by clicking the right mouse button on the place where you want to add row
   b. You can delete row(s) by selecting the rows to be deleted and clicking right mouse button, and select Remove row(s)
   c. When you have done the editing, press **Store** in the toolbar

4. If there is a need to save the mask file with different name, select **Mask save**, browse for location and name the file as wanted

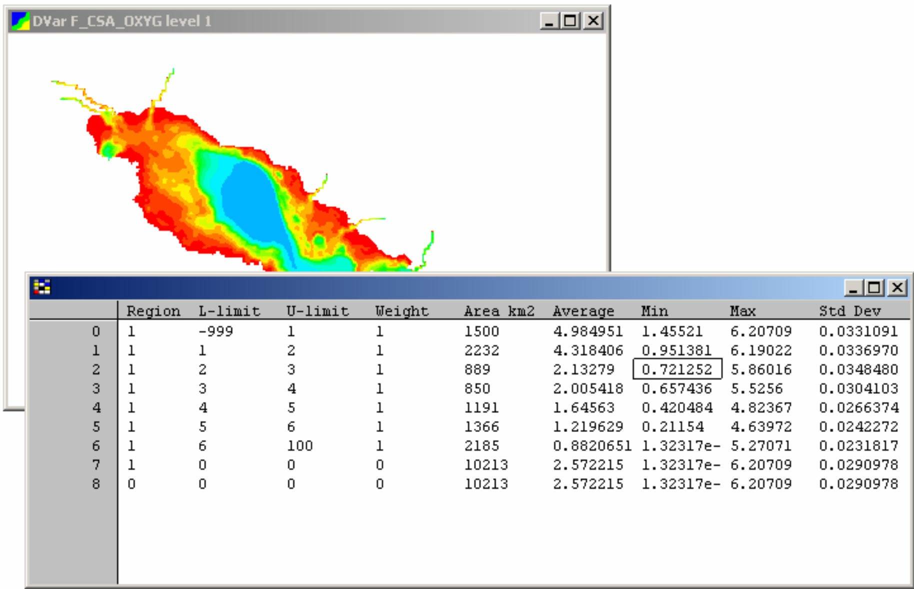
5. After the mask file is loaded, the Mask variable need to be selected from the drop-down menu in the “Compute setup” window. Click on the arrow and select the variable your mask is based on:
    a. Landuse based zoning select F_LU



    b. Elevation based zoning select F_GRELEV



6. Now you can follow the instructions from **Draw and compute** section of this chapter to select the variable and draw that
7. As a result you get the picture as in normal analysis but also the table which describes the parameter you are analysing zone by zone as presented below.
    a. The first rows present the results zone by zone
    b. the final 2 rows (only one region in use) present the total for the whole modelled area.
    c. The values of Area of the zone and Average, minimum, maximum and standard deviation of the parameter are presented.



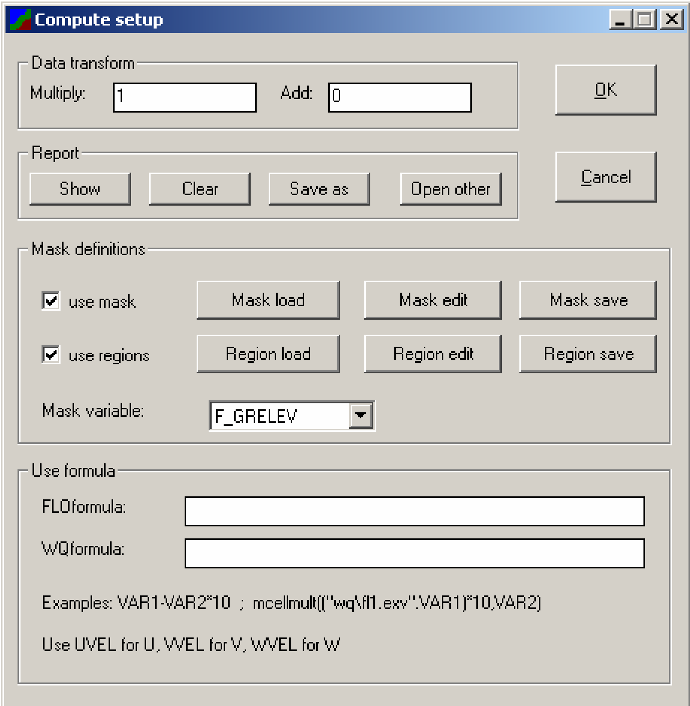
	Region	L-limit	U-limit	Weight	Area km2	Average	Min	Max	Std Dev
0	1	-999	1	1	1500	4.984951	1.45521	6.20709	0.0331091
1	1	1	2	1	2232	4.318406	0.951381	6.19022	0.0336970
2	1	2	3	1	889	2.13279	0.721252	5.86016	0.0348480
3	1	3	4	1	850	2.005418	0.657436	5.5256	0.0304103
4	1	4	5	1	1191	1.64563	0.420484	4.82367	0.0266374
5	1	5	6	1	1366	1.219629	0.21154	4.63972	0.0242272
6	1	6	100	1	2185	0.8820651	1.32317e-	5.27071	0.0231817
7	1	0	0	0	10213	2.572215	1.32317e-	6.20709	0.0290978
8	0	0	0	0	10213	2.572215	1.32317e-	6.20709	0.0290978

The formula calculating and mask tool cannot be used at the same time. However, the mask and region tools can be used parallel.

13.2.13 Using the region in “Set compute”

Addition to mask or separately from it the model area can be divided to regions based on coordinates given by user. To use the region option, please follow the instruction below (here the option where both, mask and region, are used):

1. Click set compute on the main analysis window
2. Activate the region by ticking the **Use region** box



3. Load the mask by clicking the **Region load** button and select the mask file you want to use (*.rgn)

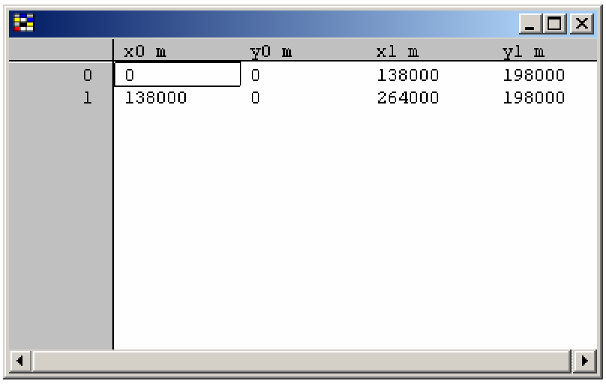
If you don’t have any *.rgn file available, you can create new one following the steps below:

- Create an empty txt-file with e.g. notepad
- First line defines the matrix’s dimensions (n*4), following lines include the data as shown below where example of 2*4 matrix of the region definition

  matrix(2 4
  0 0 138000 198000
  138000 0 264000 198000)

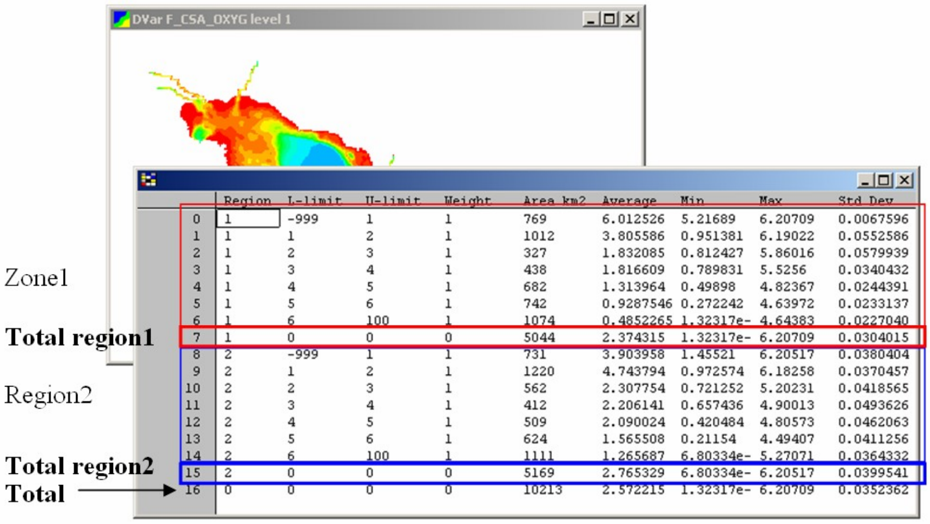
- Save the file as *.rgn (e.g. northsouth.rgn)
- Load the file as defined in step 2

4. To see the mask file click the **Region edit** button and the following window appears. Here the user can edit the region boundaries by setting the grid coordinates (meters from the origo) for it.



   a. You can add rows by clicking the right mouse button on the place where you want to add row
   b. You can delete row(s) by selecting the rows to be deleted and clicking right mouse button, and select Remove row(s)
   c. When you have done the editing, press **Store** in the toolbar to save the edits

5. If there is a need to save the region file with different name, select **Region save**, browse for location and name the file as wanted
6. Now the setup is ready and you can accept the setup by clicking **OK** button on the window and that will return you to “Field draw and compute window”
7. Now you can follow the instructions from **Draw and compute** section of this chapter to select the variable and draw that
8. As a result you get the picture as in normal analysis but also the table which describes the parameter by regions you selected.
   a. Here both, region and mask load were used and thus the mask results are divided for those two region
   b. The first rows present the results zone by zone for he region 1 and the final row for that region present the total
   c. The final row where region number is 0, presents the total for the whole modelled area
   d. The values of Area of the zone and Average, minimum, maximum and standard deviation of the parameter are presented



	Region	L-limit	U-limit	Weight	Area km2	Average	Min	Max	Std Dev
0	1	-999	1	1	769	6.012526	5.21689	6.20709	0.0067596
1	1	1	2	1	1012	3.805586	0.951381	6.19022	0.0552586
2	1	2	3	1	327	1.832085	0.812427	5.86016	0.0579939
3	1	3	4	1	438	1.816609	0.789831	5.5256	0.0340432
4	1	4	5	1	682	1.313964	0.49898	4.82367	0.0244391
5	1	5	6	1	742	0.9287546	0.272242	4.63972	0.0233137
6	1	6	100	1	1074	0.4852265	1.32317e-	4.64383	0.0227040
7	1	0	0	0	5044	2.374315	1.32317e-	6.20709	0.0304015
8	2	-999	1	1	731	3.903958	1.45521	6.20517	0.0380404
9	2	1	2	1	1220	4.743794	0.972574	6.18258	0.0370457
10	2	2	3	1	562	2.307754	0.721252	5.20231	0.0418565
11	2	3	4	1	412	2.206141	0.657436	4.90013	0.0493626
12	2	4	5	1	509	2.090024	0.420484	4.80573	0.0462063
13	2	5	6	1	624	1.565508	0.21154	4.49407	0.0411256
14	2	6	100	1	1111	1.265687	6.80334e-	5.27071	0.0364332
15	2	0	0	0	5169	2.765329	6.80334e-	6.20517	0.0399541
16	0	0	0	0	10213	2.572215	1.32317e-	6.20709	0.0352362

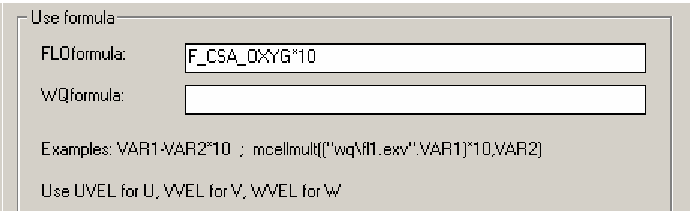
The formula calculating and mask tool cannot be used at the same time. However, the mask and region tools can be used parallel, but also separately.

13.2.14 Using the formula in “Set compute”

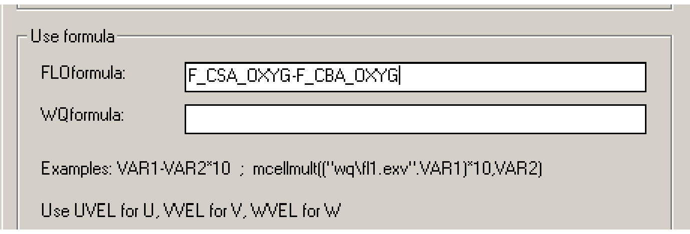
One option in Compute setup is to use formula for analysis.

With formula the user is able to e.g.

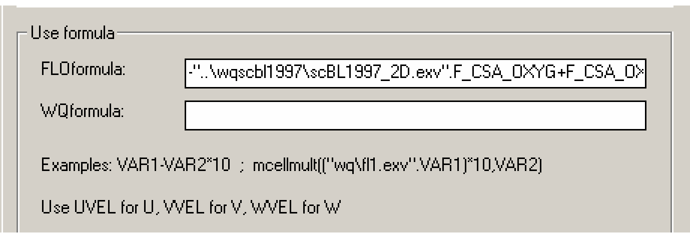
- multiply the results (e.g. if user wants to change the unit)



- add value to the parameter or then calculate the differences between two variables from same run (e.g. bottom and surface oxygen)

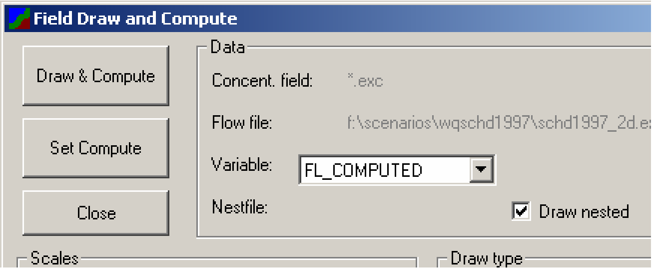


- calculate the differences between two model runs (e.g. scenario – baseline)

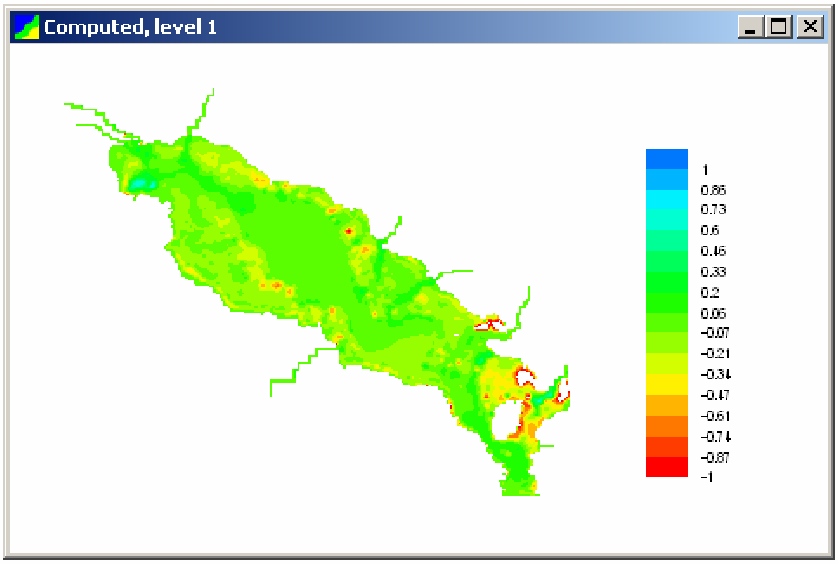


Below there are steps to use the formula

1. Write the formula to the FLOformula field
2. After the formula is written, click **OK** and the main analysis window appears
3. Select **FL_COMPUTED** for the variable and select Draw & Compute



4. The result of the formula will be drawn to as a result

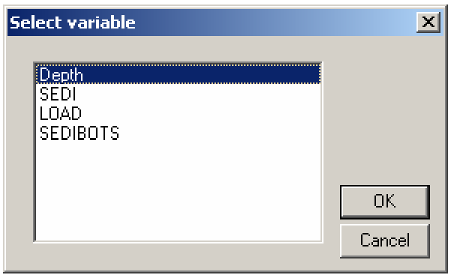


The formula calculating cannot be used at the same time with the mask and/or region tool(s).

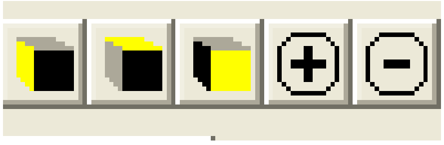
13.3 Sections

With the sections analysis tool user is able to draw fast cross section fields in any directon (top – down, west – east, south - north) and step through the water body.

Selectiong Analysis – Sections options open first a selection of a variable to be drawn:

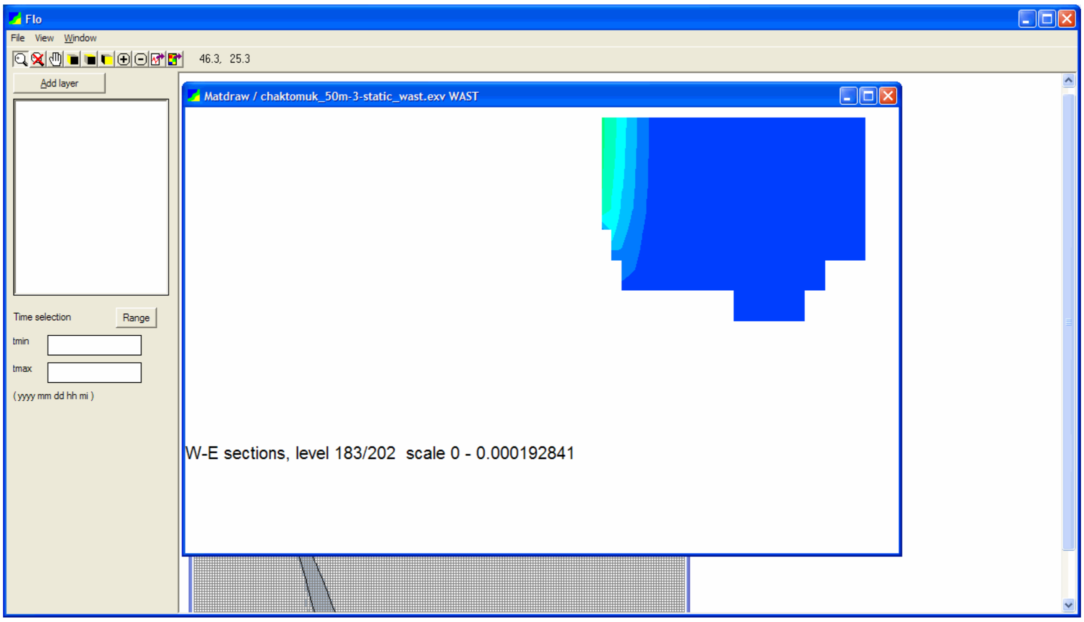


After that a drawing window with a view from top opens. User has the following options to select sections to be drawn:

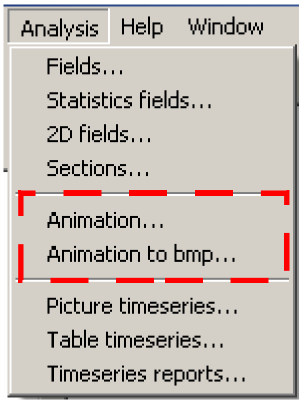


The boxes with a yellow wall show output selection direction: leftmost box select north-south cross-sections, middle box selects top view and right box selects west-east cross-sections. + - control advances and – decreases section or layer.

The resulting view is presented below. In it is a west-east cross section from row 183 out of 202 rows.



14 ANIMATION



This chapter includes the instructions how to see the stored animation and use the animation tool.

The chapter is divided to two parts:

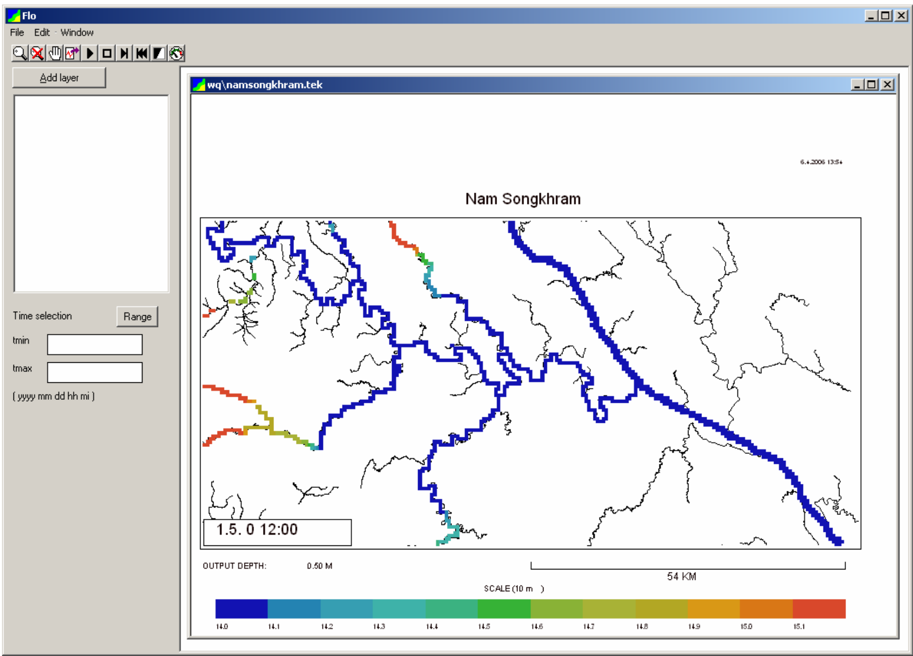
14.1 Seeing the animation

14.2 Animation to bmp-file

14.1 Seeing the animation

User is able to store the view of the running model as a TEK-animation file. The animation can be seen when selecting **Analysis – Animation…** from the main menu.

To be able to display the animation, the TEK-animation should be checked on in the Output files section of the “Animation parameters” window (Model – Animation… - Animation options) as defined in Section 10.3 – Animation options.



The animation shows the concentration of animated variable as a colour from blue to red. Animation timeseries, if defined to be shown, is visible on the right side of the picture. The black dot in the animation picture shows the timeseries point location. Date is displayed in the box on the left bottom part of the picture.

The animation output can be controlled through the toolbar shown below:

a) b) c) d) e) f) g) h) i) j)

The following functions (labelled below the toolbar) can be used:

a. Zoom in

b. Zoom out

c. Pan

d. The animation returns automatically to the beginning if the animation window is not the topmost window on the screen

e. Start animation/continue animation after it has been stopped

f. Pause the animation

g. Go one animation frame forward

h. Go to beginning of the animation

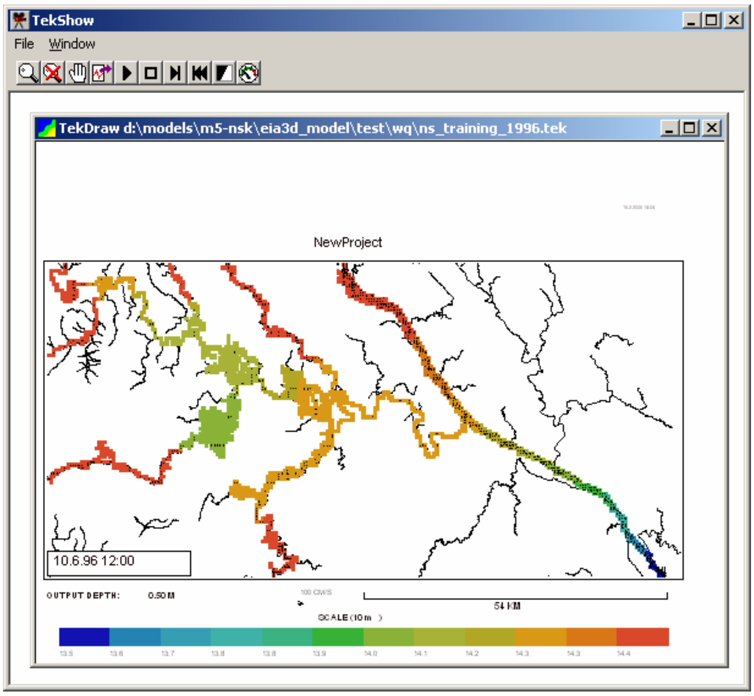
i. Change the colour of the background from white to black and vice versa

j. Set the time lag between animation frames

The animation can be started also directly from the TEK-animation file. TEK-animation files, all other output files, are stored to the Flow directory (defined in "Application setup" window under Source data – Application setup).

ns_training_feb2006.TEK	342 KB	TEK File	20/02/2006 14:20
nsk3.TEK	42,122 KB	TEK File	16/02/2006 10:10
nsk4.TEK	19,320 KB	TEK File	13/02/2006 14:58
nsk4_1.TEK	58,221 KB	TEK File	13/02/2006 09:35
nsk_new2.TEK	1,957 KB	TEK File	08/03/2006 11:57
nsk_new.TEK	50,676 KB	TEK File	14/02/2006 13:26

When opening the TEK-animation file (double click the file name), the following "TekShow" window appears.



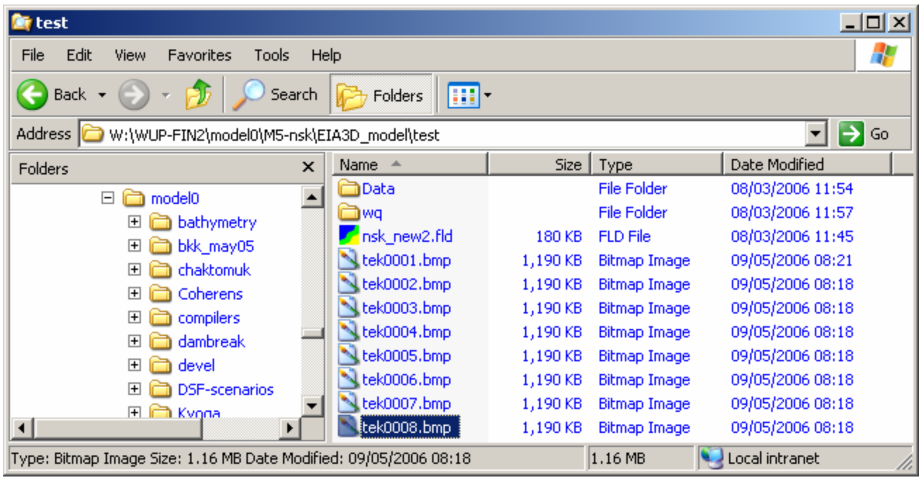
In "TekShow" window user has the same tools to manage the animation as described above.

> The animation returns automatically to the beginning if the animation window is not the topmost window on the screen.

14.2 Animation to bmp-file

User is able to save each animation frame as a separate *.bmp (Bitmap Image) file by selecting the **Analysis – Animation to bmp…** from the main menu. The similar animation window and tools are available for the user as defined in section 14.1 "Seeing the animation".

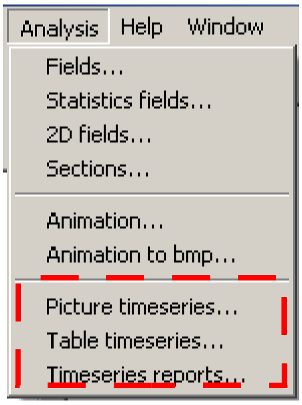
Each frame the user will go through will be saved as *.bmp to the same folder where the *.fld is located as shown below.



The *.bmp files can be further used by exporting them for different types of animation files (*.gif, *.avi, etc). This can be done by e.g. Jasc Animation shop by Jasc software.

One *.bmp file can be rather large (0.5-2.0 MB) and thus, user should be careful not to exceed the capacity of the hard disk when transferring hundreds of frames to *.bmp files.

15 TIMESERIES OUTPUT



This chapter includes the instructions how to present the timeseries data and use the timeseries analysis tool.

The chapter is divided to four parts:

15.1 Picture timeseries

15.2 Table timeseries

15.3 Timeseries reports

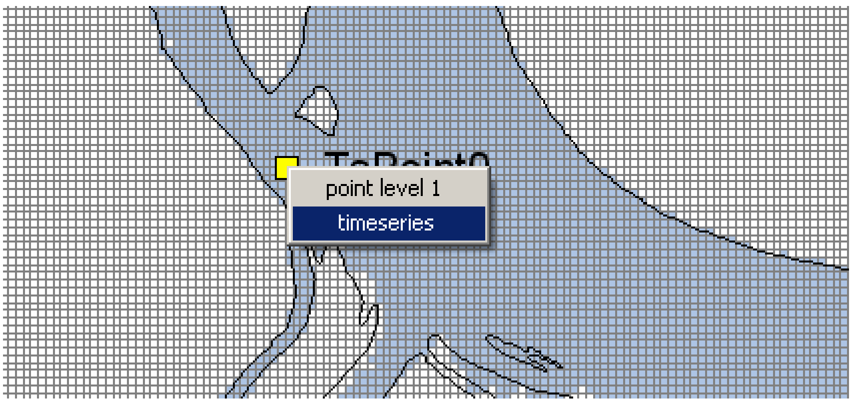
15.4 Timeseries analysis tools

15.1 Picture timeseries

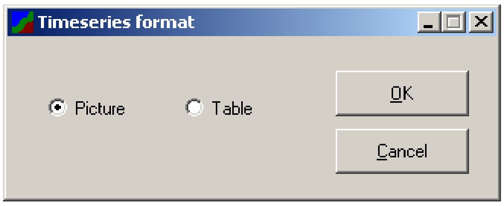
15.1.1 Picture timeseries parameters

By selecting **Analysis – Picture timeseries…** user is able to use plot the results of selected parameters for all the timeseries points defined in **Model – Timeseries – Points/Options** (see Section 10.4 – Timeseries handling and options for more information of how to add new timeseries point, and how to select the timeseries output variables).

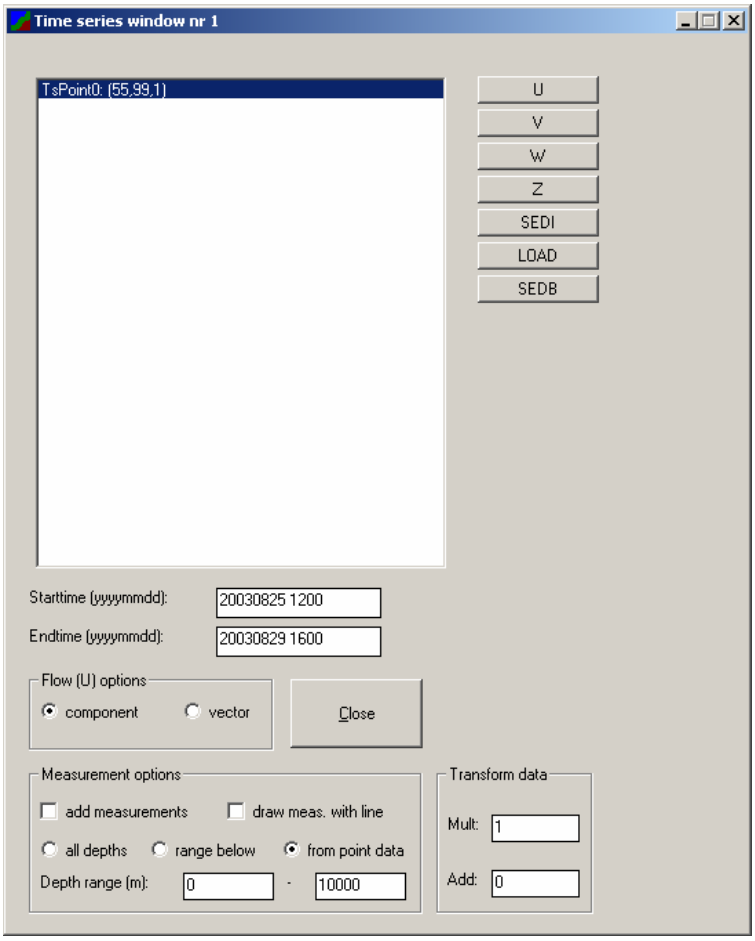
Other option to start plotting the results of selected timeseries point is to click left mouse button on top of the TsPoint in the main model window and select **timeseries** from the menu list



The **Timeseries format** (below) window appears. To start plotting the results, check **Picture** and press **OK**.



By selecting either the **Analysis – Picture timeseries…** or **Picture** in **Timeseries format** window, the **Timeseries window nr. #** appears as presented below.

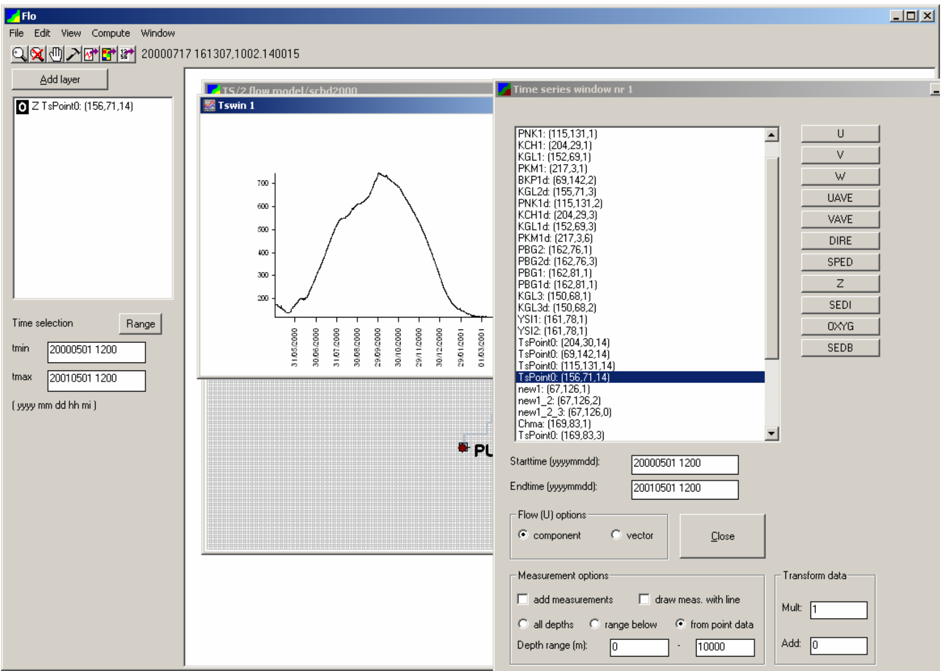


The **Timeseries window nr. #** window is divided to six categories:

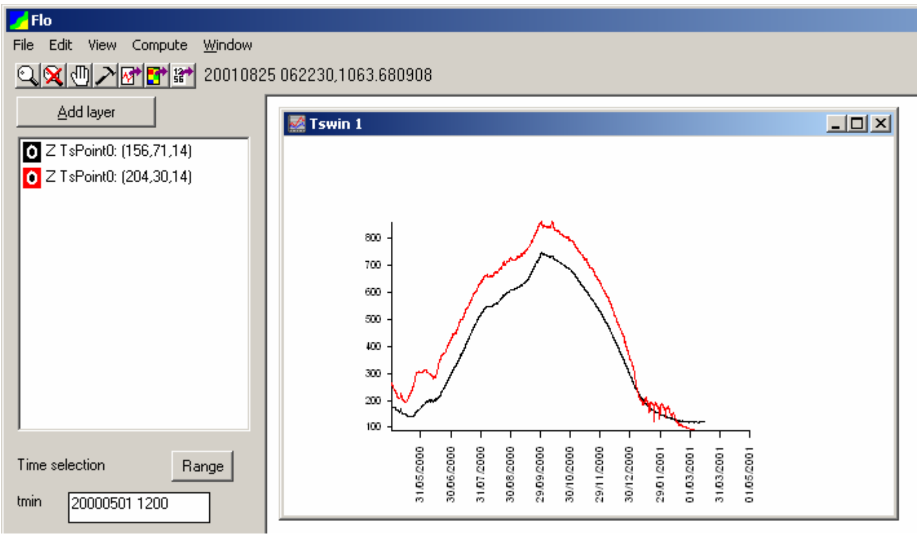
- List of timeseries points – on left part of the window there is a list of the available timeseries points (Ts point)
- Variables – on the right from the list of the Ts points, the variables selected by user (see Section 10.4 – Timeseries handling and options) for analysis are listed. The parameters' abbreviations and units are listed in Section 27.2 – Output selection symbols (in GUI and graphics files).
- Start and end times - user can define the analysed time period
- Flow (U) options – options for flow
- Measurement options – if the timeserie point includes measurement, the measured values can be drawn to the same figure
- Transform data – the data can be either multiplied by certain factor (e.g. change of unit) or user can add certain value for each dataset (e.g. change reference level)

Follow the steps below to plot the timeseries picture

1. Select the timeseries point from the “List of timeseries” you want to use the information from
2. Select the start time and end time for the data plotting. Default time period is the whole simulated time
3. Set up the other options (Flow, Measurement, Transform) if needed
4. Select the parameter to be plotted (see Section 27.2 – Output selection symbols (in GUI and graphics files) for more information of the parameters) by clicking the parameter button
5. The timeseries picture appears on the main application window



6. User can compare the results in different locations/depths by selecting another timeseries point and selecting the same variable to be plotted

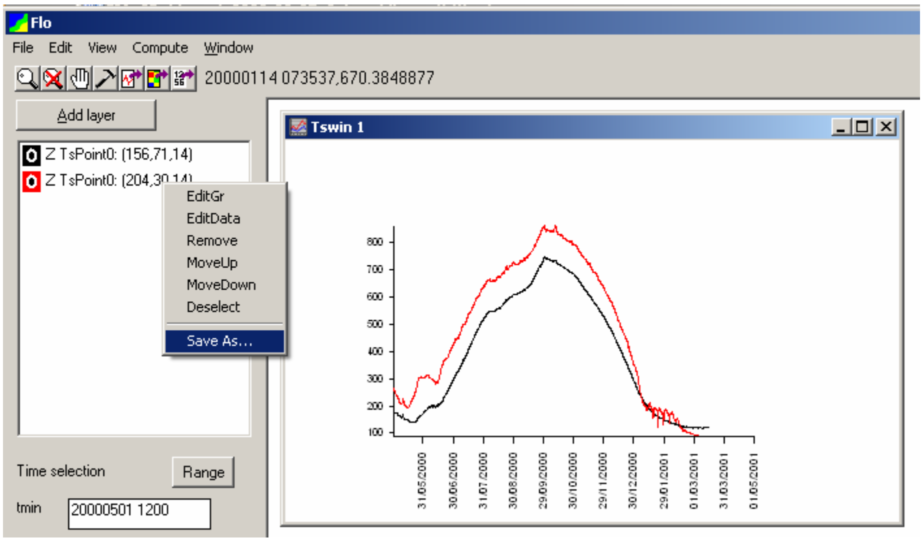


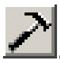
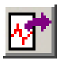
7. More about the timeseries picture management can be read from the following chapter.

15.1.2 Timeseries picture management

The timeseries drawn in the picture are listed in the small window on the left.

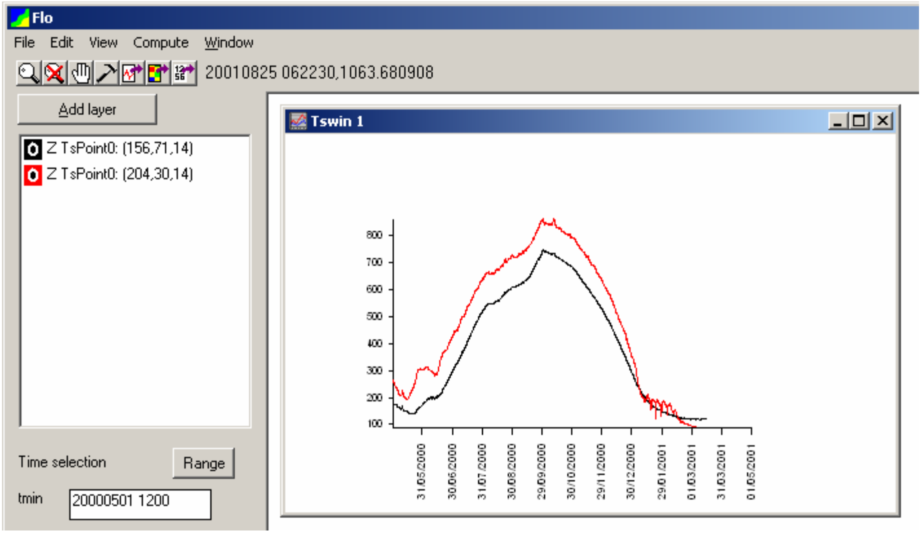
1. The timeseries can be deactivated and activated by clicking the square on the left of the timeseries name. The circle in the middle of the square tells if the timeseries is active or not. Only active timeseries are shown in the picture.
2. To select a timeseries click the left mouse button on top of the timeseries name so that the background turns grey.
3. To save the timeseries, select the correct timeseries by clicking that timeseries in the window in the left and press the right mouse button. Select **Save as** from the menu, give a name to the timeseries and press **Save**.



4. To manage the timeseries select the correct timeseries by clicking that timeseries in the window in the left and press the right mouse button. From hear he timeseries can be edited (**Edit**), removed entirely (**Remove)** and their order can be changed (**MoveUp** and **MoveDown)**.
5. The timeseries window can be managed using the timeseries window tools, which are described in following section. The appearance of the timeseries window (line colors and types, axis etc) can be modified using the **Picture properties** tool .
6. The picture can be saved as an image by clicking the **copy as metafile** tool  while the correct picture is activated. The picture can then be pasted to appropriate application (for ex. MS Word) by selecting paste in that application.

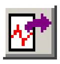
15.1.3 Timeseries window tools

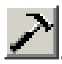
Timeseries window shows the output data timeseries as picture.

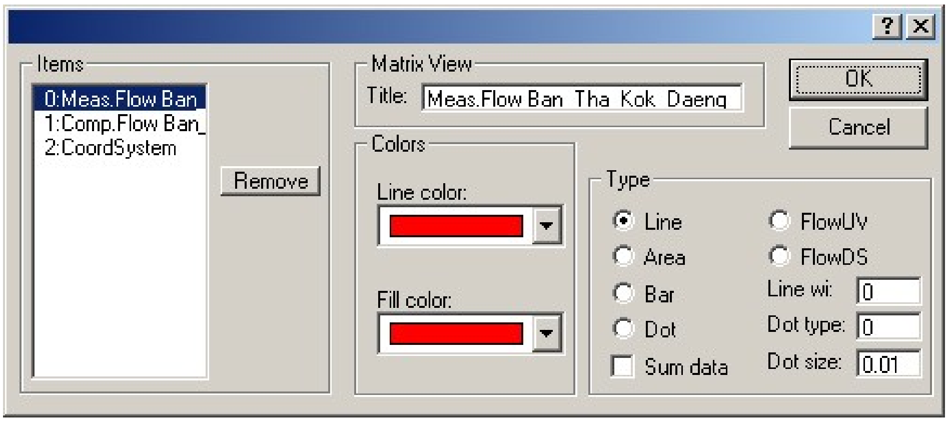


The picture handling tools are available when the timeseries window is active. The tools for picture handling are

The first three tools from left a are the same as in EIA 3D model main window toolbars (see Section 4.2.2 – Tools menu bar). The functions of the other tools are

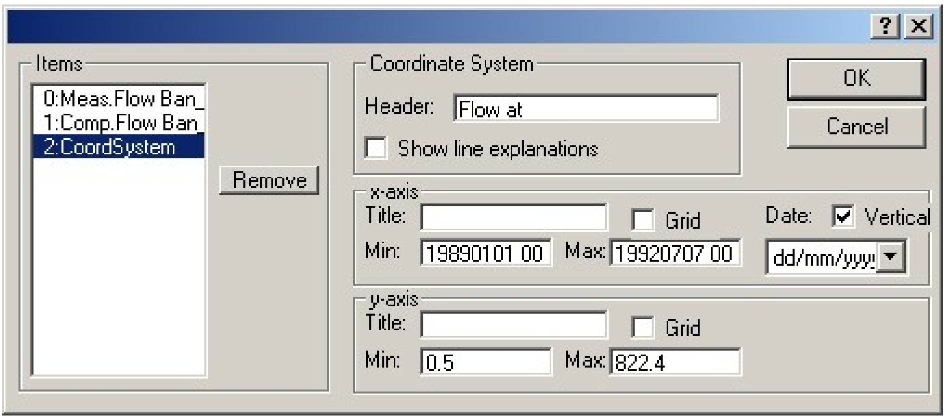
Tool	Function
	Picture properties. Displays a format window, where picture properties (line colour, type, name etc.) can be modified.
	Copy as metafile. Copy the picture to the clip-board as a metafile.
	Copy as bitmap. Copy the picture to the clip board as a bitmap.
	Copy as text data. Copies the picture data in text format to clipboard. Works only for timeseries pictures. Can be used, for example, to transfer picture data to spreadsheet.

Picture properties can be changed with the **Picture properties** tool . A format window, where the outlook of the picture can be managed will open.

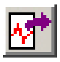


The window shows all the timeseries drawn in the picture in the window on the left. The timeseries being managed can be changed from the window in the left hand corner of the main window. The colour of the timeseries can be changed from Colors **Line Color** and **Fill Color** (for dots and areas). The timeseries can be drawn as line, bar, column or line and the sizes and widths of these can be changed from the right hand corner of the window.

To change the coordinates of the picture choose **CoordSystem** from the list.



1. The Header of the picture can be given from **Header**.
2. The titles of the x-axis and y-axis can be given from **Title**.
3. The minimum and maximum values of the x- and y-axis of the picture can also be changed from **Min** and **Max**.

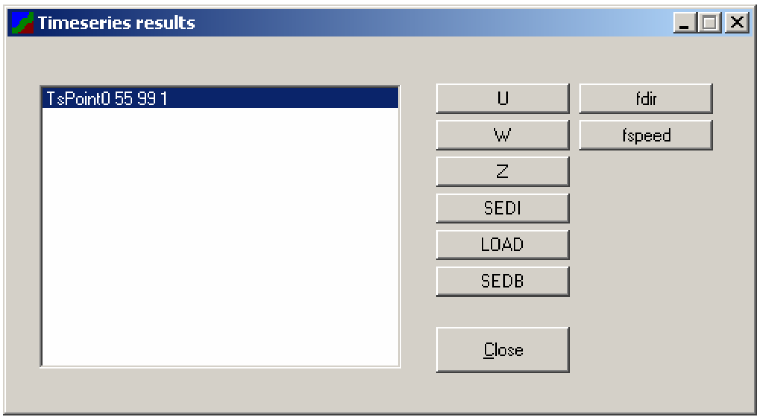
The picture can be saved as an image with the **copy as metafile** tool  or copy as bitmap tool . While the correct picture is activated press the **copy as metafile** button. Then go to the application where you want the picture to be (for example a Microsoft Word document) and select Paste. The picture will appear in the application.

15.2 Table timeseries

15.2.1 Table parameters

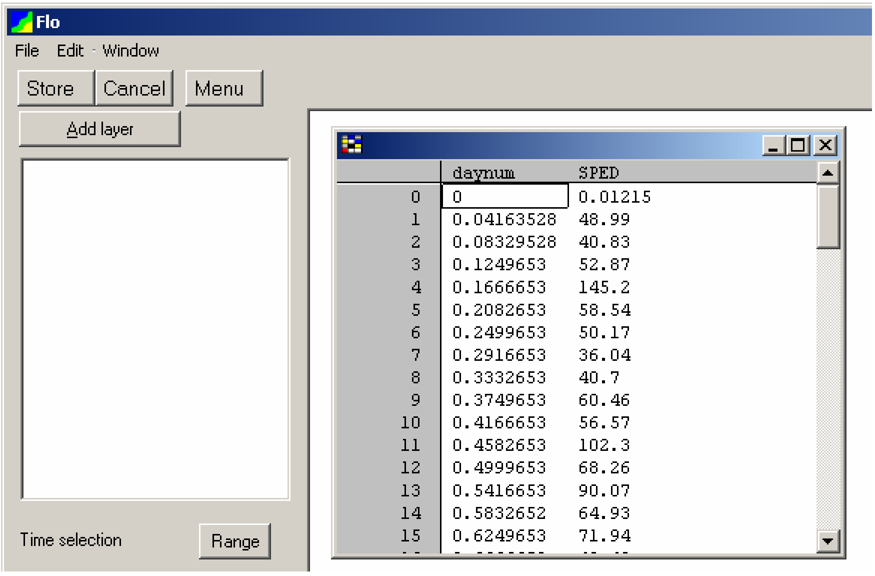
By selecting **Analysis – Picture timeseries…** user is able to use see table of the results of selected variables for all the timeseries points defined in **Model – Timeseries – Points/Options** (see Section 10.4 – Timeseries handling and options for more information).

The **Timeseries results** window is presented below.



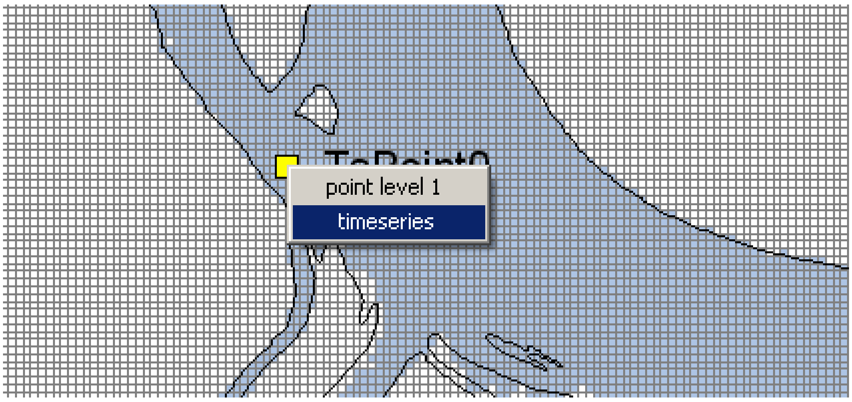
The list of the timeseries points is on the left and list of the stored variables on the right. Follow the steps below to see the table of the wanted TsPoint and variable by using **Timeseries results** window.

1. Select the timeseries point from the list
2. Click the variable you want to include into the table and the table appears as illustrated below.

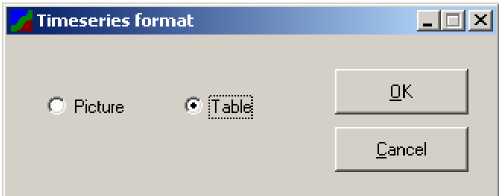


3. The explanation and unit for each variable can be found from Section 27.2 – Output selection symbols (in GUI and graphics files)
4. The table management is explained in the following section

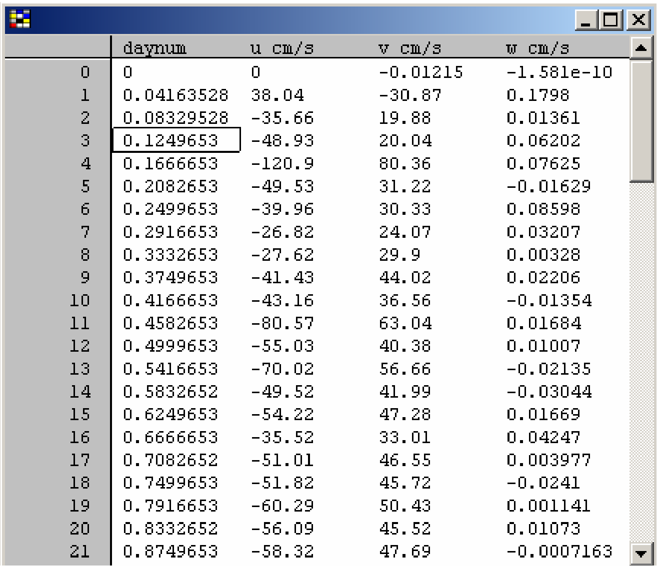
Other option to see the timeseries results in table format is to click left mouse button on top of the TsPoint in the main model window and select timeseries



The **Timeseries format** (below) window appears. To see the table of the variable defined (Section 10.4 – Timeseries handling and options) the results check **Table** and press **OK**.



The following table appears



	daynum	u cm/s	v cm/s	w cm/s
0	0	0	-0.01215	-1.581e-10
1	0.04163528	38.04	-30.87	0.1798
2	0.08329528	-35.66	19.88	0.01361
3	0.1249653	-48.93	20.04	0.06202
4	0.1666653	-120.9	80.36	0.07625
5	0.2082653	-49.53	31.22	-0.01629
6	0.2499653	-39.96	30.33	0.08598
7	0.2916653	-26.82	24.07	0.03207
8	0.3332653	-27.62	29.9	0.00328
9	0.3749653	-41.43	44.02	0.02206
10	0.4166653	-43.16	36.56	-0.01354
11	0.4582653	-80.57	63.04	0.01684
12	0.4999653	-55.03	40.38	0.01007
13	0.5416653	-70.02	56.66	-0.02135
14	0.5832652	-49.52	41.99	-0.03044
15	0.6249653	-54.22	47.28	0.01669
16	0.6666653	-35.52	33.01	0.04247
17	0.7082652	-51.01	46.55	0.003977
18	0.7499653	-51.82	45.72	-0.0241
19	0.7916653	-60.29	50.43	0.001141
20	0.8332652	-56.09	45.52	0.01073
21	0.8749653	-58.32	47.69	-0.0007163

Where the first column is the time (either date if those used for calculation or then time started from the 0). The variable columns follow the time.

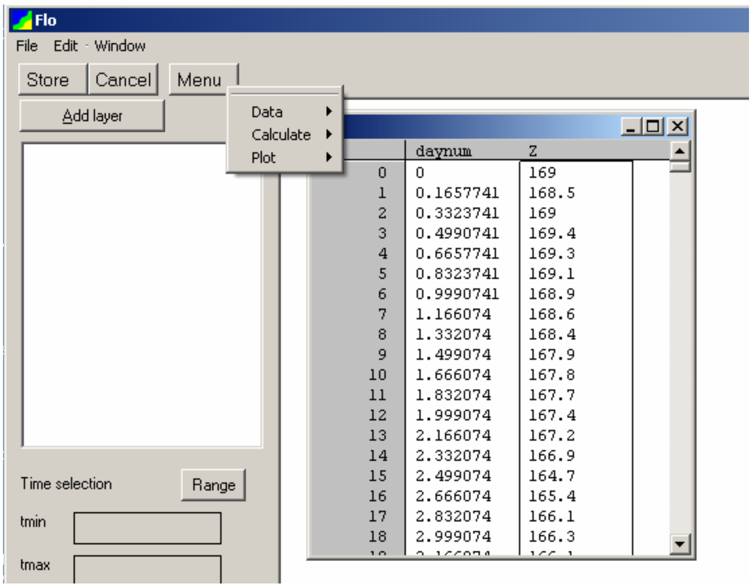
15.2.2 Table management

When the table is activated the following buttons appears on the toolbar

Store – save the made changes in the table and closes the table window

Cancel – closes the table window without saving the possibly made edits

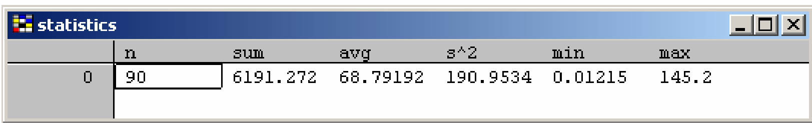
Menu – opens the menu to operate with the table data as illustrated below



Menu/Data – user has options to do the following actions:

- Create matrix: creates an empty matrix
- Write to file: write the table to txt or txd file

Menu/Calculate/Statistics – user can calculate basic statistics for the selected table variable



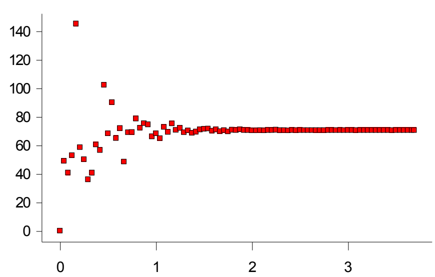
statistics

	n	sum	avg	s^2	min	max
0	90	6191.272	68.79192	190.9534	0.01215	145.2

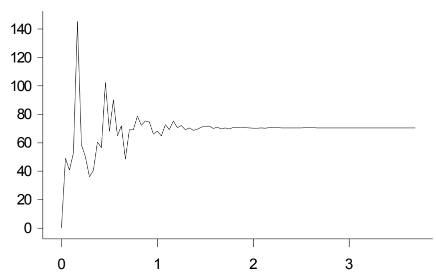
Menu/Plot – user has several options to plot the selected data from the table

- XY dot
- XY line
- Area
- Line
- Bar

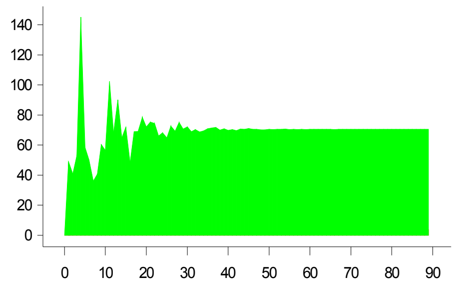
Below some the example plots have been presented:



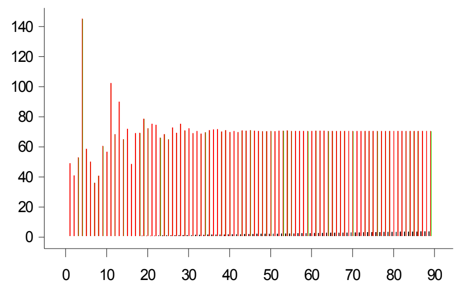
XY dot



XY line



Area

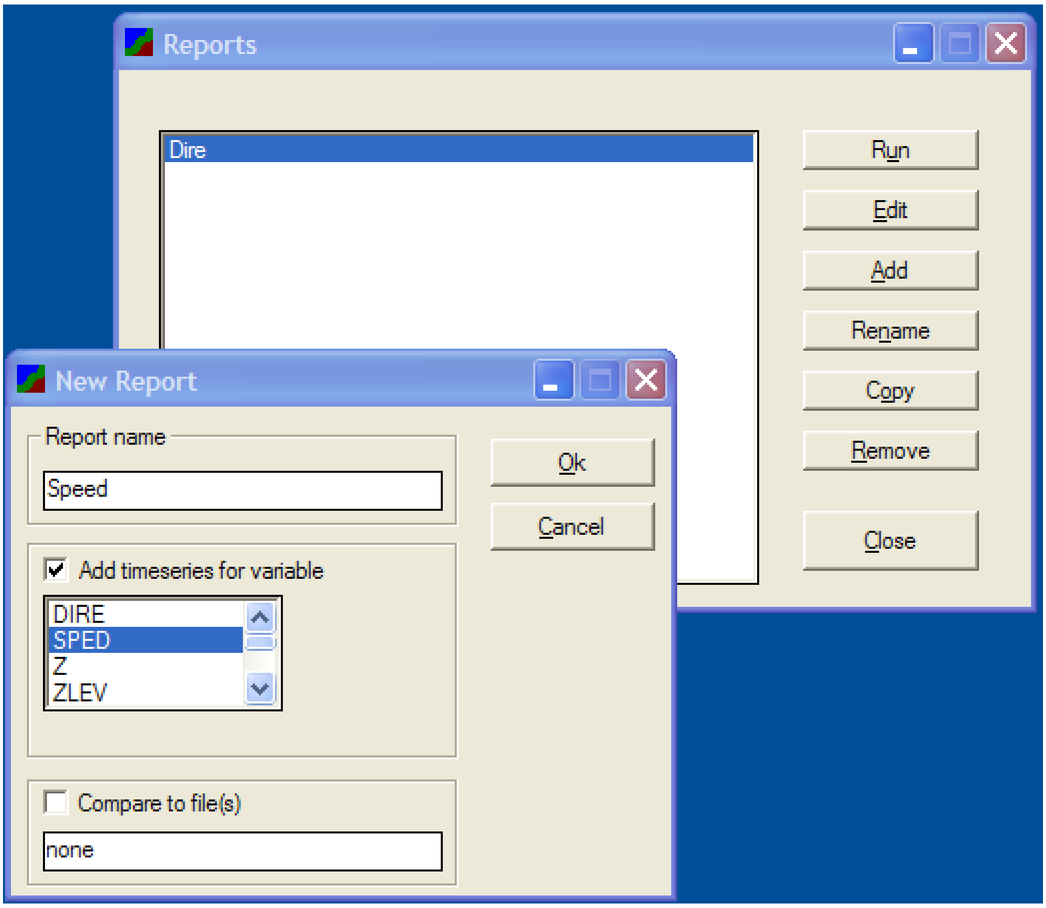


Bar

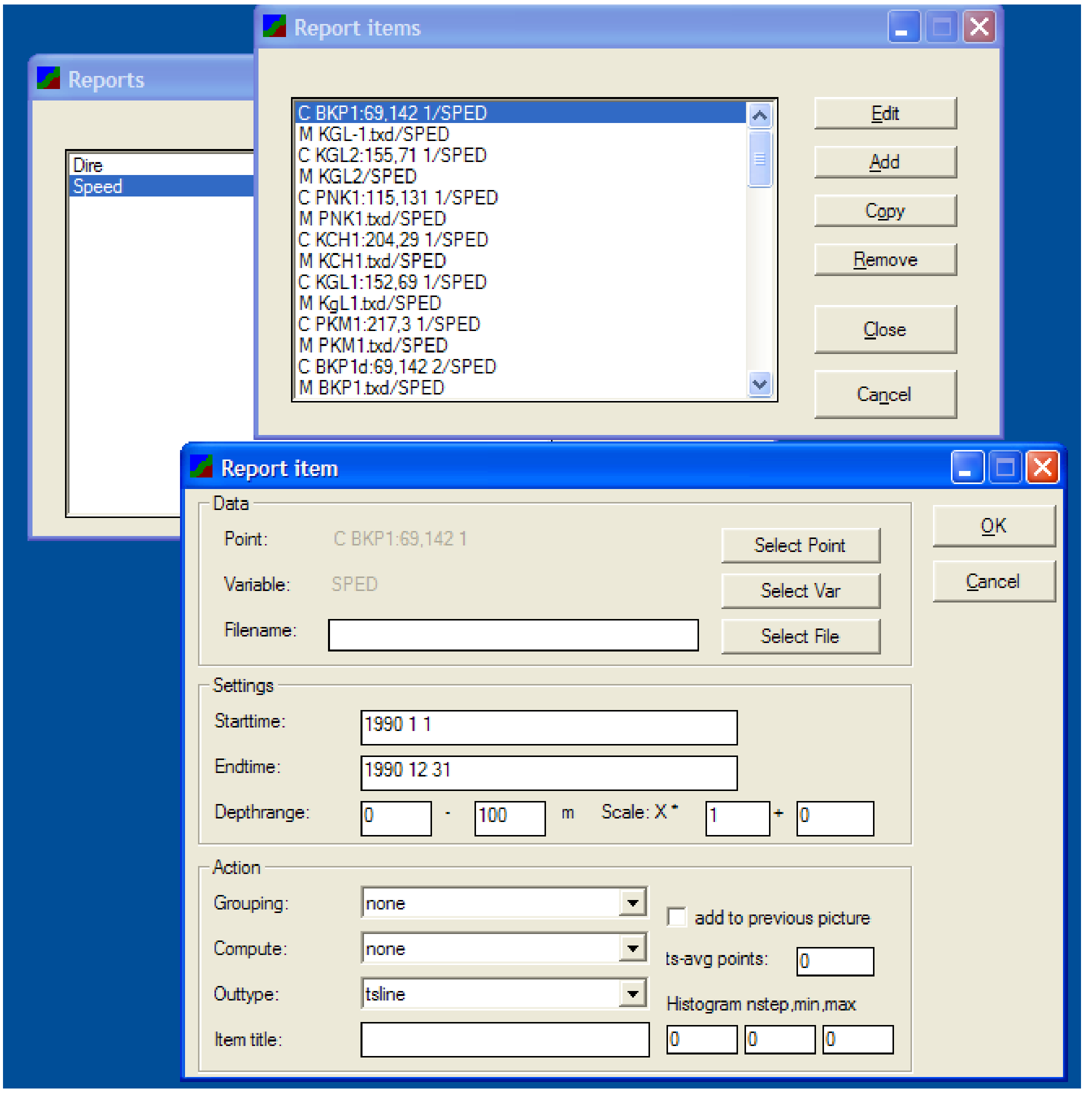
15.3 Timeseries reports

Timeseries reports are a way to define and draw a large number of time series automatically. Typically this can be used as a help to output timeseries for repeated scenario runs, calibration, or periodical reporting.

When user selects **Analysis – Timeseries results**, a Reports window is opened. After selecting **Add**-button, user can select variable to be drawn:



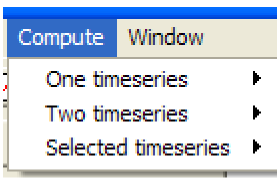
By default all timeseries points are selected for drawing. User can remove unneccessary points and define drawing options for each point separately:



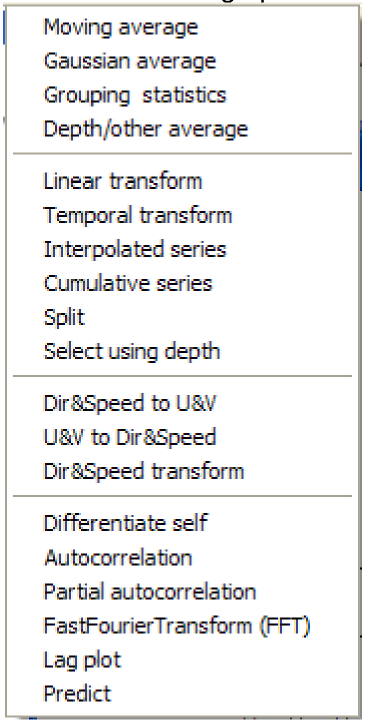
At the moment setting of reporting tool options is not totally automated, and user may need to specify directories and variables by hand.

15.4 Timeseries analysis tools

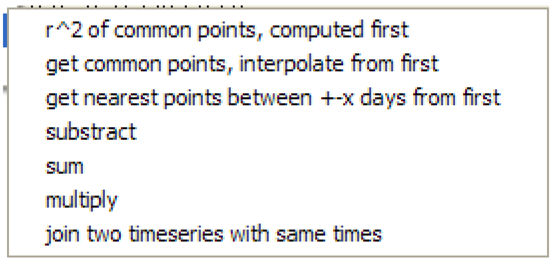
When one or more timeseries have been drawn and a timeseries window is selected, users has following options available for timeseries analysis:



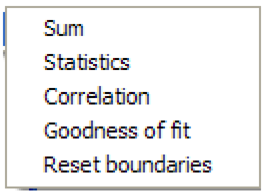
One timeseries analysis contains following options:



Two timeseries have following options

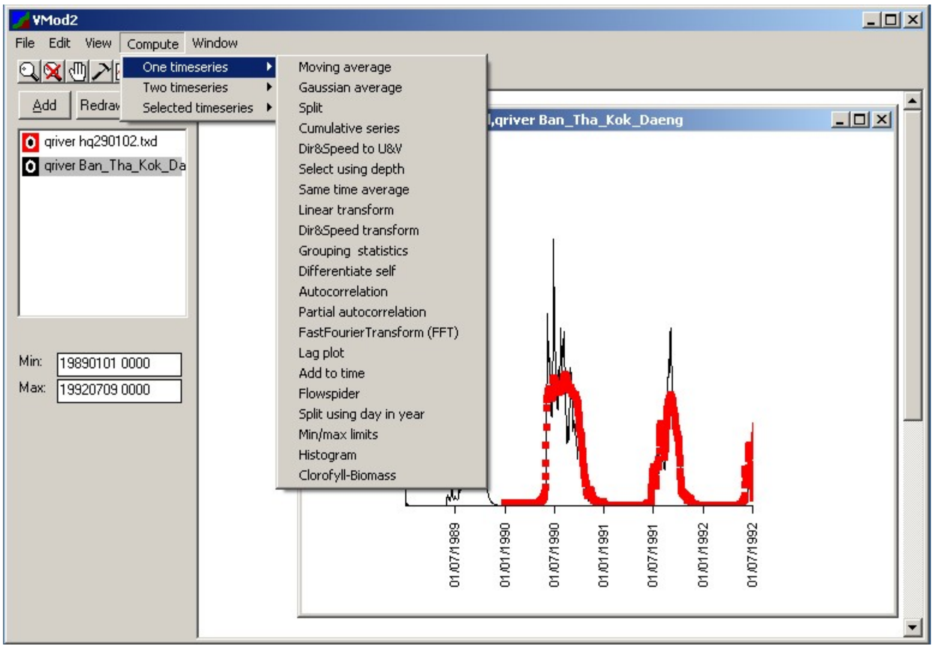


Selected timeseries options has following options:



Comparison tests

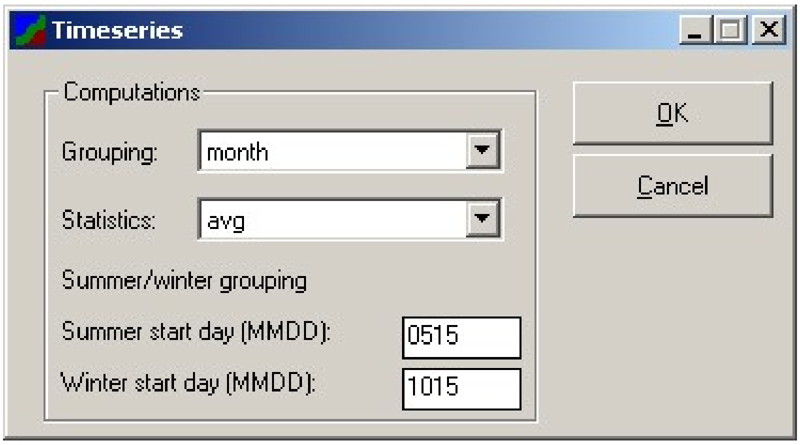
Different tests and comparisons can be performed for the results while in the timeseries result window with the options of the **Compute** menu. There are several options and only the most commonly used one's are described below.



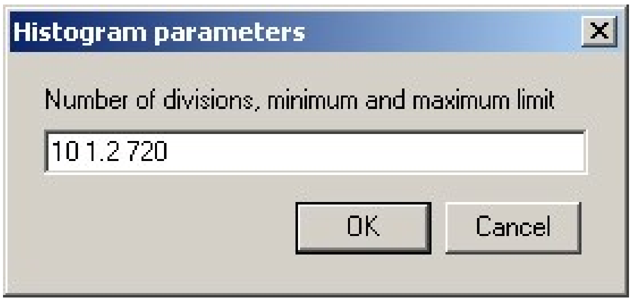
The timeseries that is evaluated has to be chosen by clicking the name of the timeseries on the left window (press control to select more than one). The timeseries is selected when it appears in grey background.

For **Compute - One timeseries** the actions available include for example

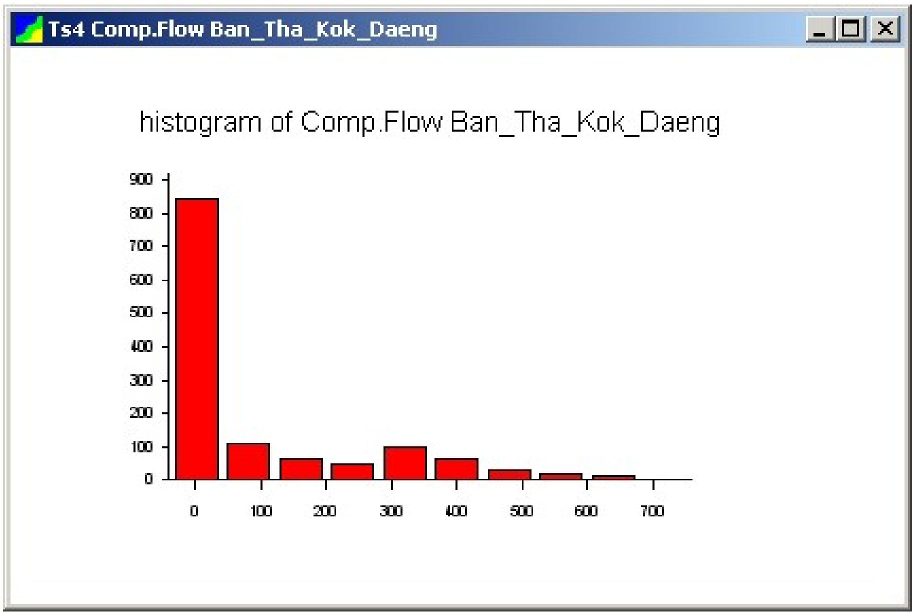
1. **Cumulative series**. This action produces a new timeseries in the picture showing the cumulative value of the timeseries chosen.
2. **Grouping statistics.** With this action you can calculate for example monthly and yearly averages, maximums, and minimums. You can also group the timeseries to winter and summer (or this action can also be used to group to dry season and wet season) and change the start dates of these periods. Choose the time period and statistics you want to evaluate from the dropdown menus in **Grouping** and in **Statistics**. Then press **OK**. The action will produce a new timeseries in the old picture.



3. **Histogram**. Produces a histogram of the selected timeseries. In the Histogram parameters window opening, the number of divisions, minimum and maximum limit can be changed if necessary.



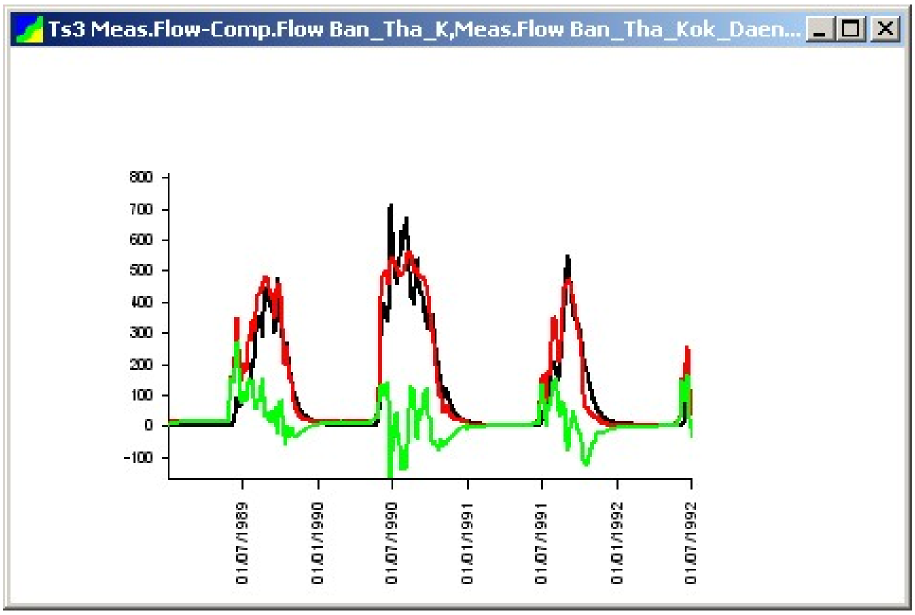
a. The histogram will be drawn in a separate window. The properties (colors, titles etc.) of the histogram picture can be managed with the picture properties tools in the same way as for timeseries pictures.



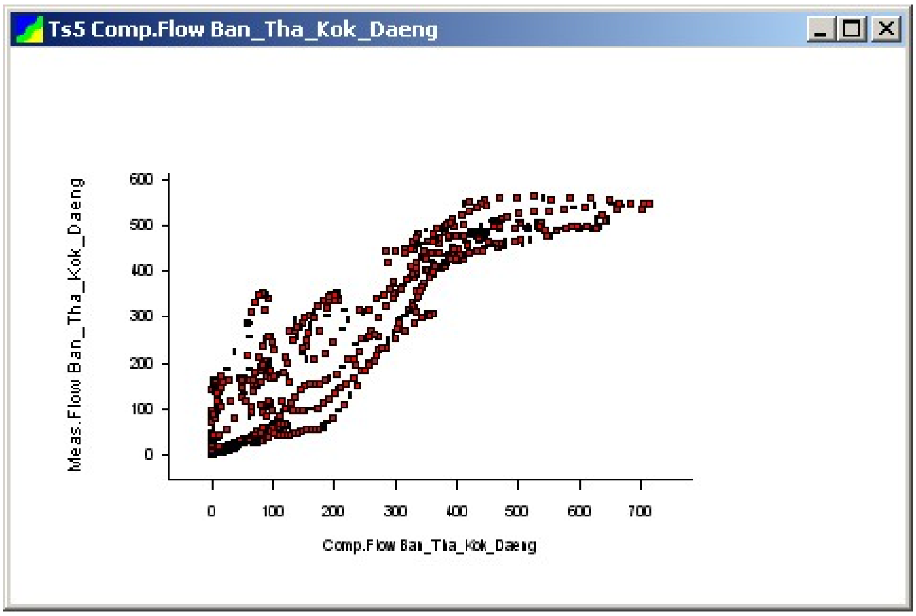
For **Compute - Two timeseries** the options are for example

1. **r^2 of common points**. Calculates the r2 measure of fit between the two timeseries chosen in a result window. The r2 measure of fit can be used to evaluate the model performance when it is used between the measured and the calculated series. The r2 fit will appear in a TsComputeReport window.

2. **subtract.** Subtracts the second timeseries from the first and produces a new timeseries in the picture with this subtraction. Can be used to evaluate the differences between the two timeseries.



3. **x-y plot**. Plots the two timeseries selected in an x-y plot in a separate window.

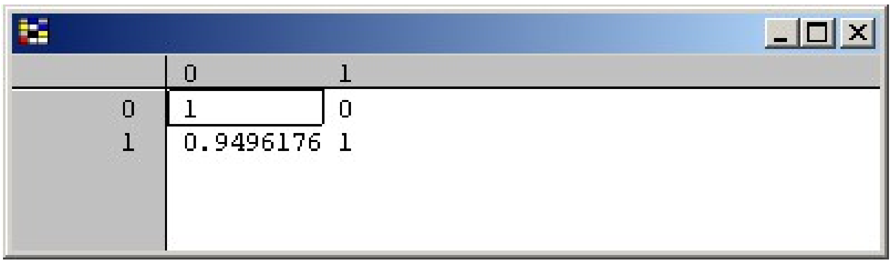


For **Compute - Selected timeseries** (one, two or more)

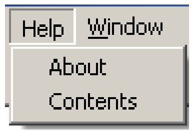
4. **Statistics.** Calculates the statistics (average, standard deviation, minimum, maximum) of the selected timeseries in a separate results window.

	n	sum	avg	std	min	max
0	1283	162321.3	126.517	177.0243	0.5	562
1	1283	143924.9	112.1784	168.9311	1.169	716.9

5. **Correlations.** Calculates the correlation between the selected timeseries in a separate result window.


0
1
0
1
0
1
0.9496176
1

16 USING HELP



This chapter includes the instructions how to use the help in the model.

The chapter is divided to two parts:

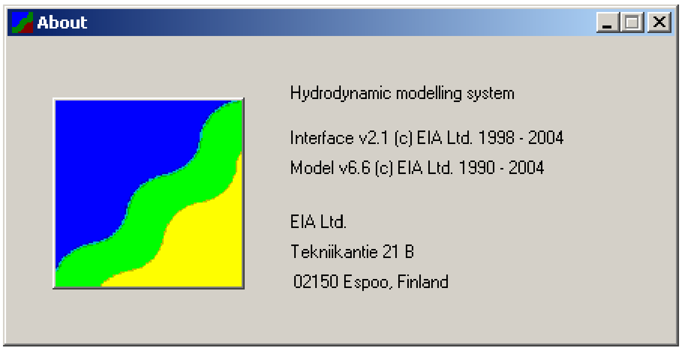
16.1 About

16.2 Help in model software

16.3 On-line help

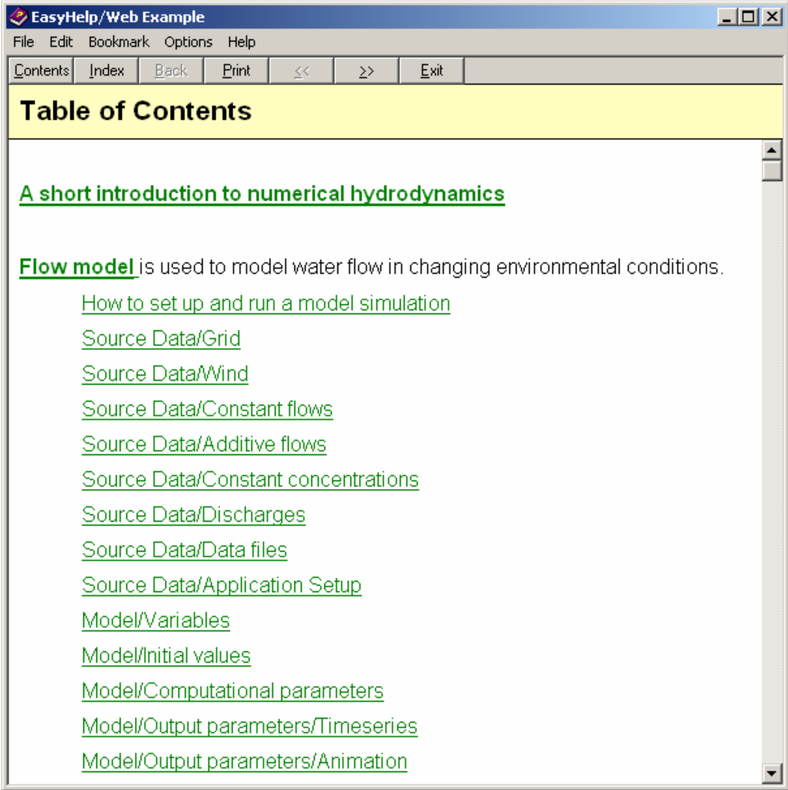
16.1 About

By clicking the **Help – About** user is able to see the version of the EIA model system

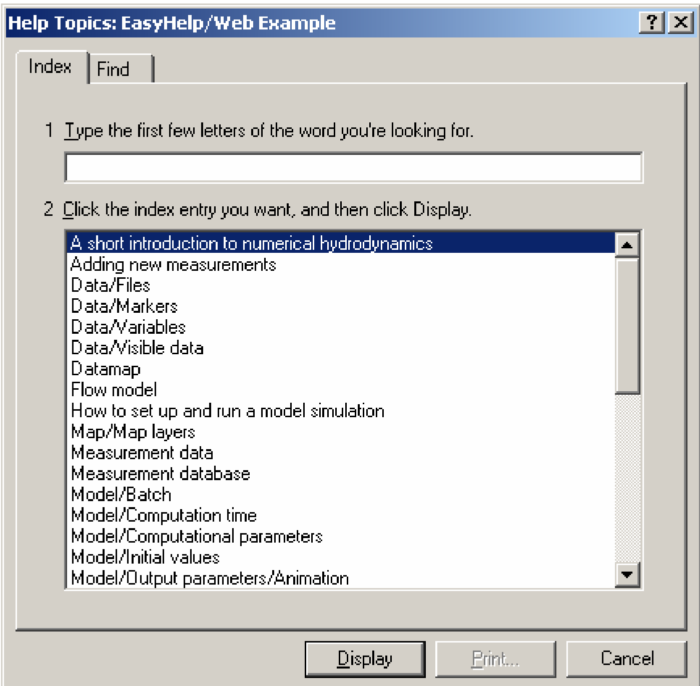


16.2 Help in model software

User can open the contents of the help system built into the model by selecting **Help – Contents**. The following window appears. User is able to follow the link to the issue he wants help for.



If user wants to use Index or Find she/he needs to click Index on the toolbar in EasyHelp window and following Index / Find window appears.



16.3 On-line help

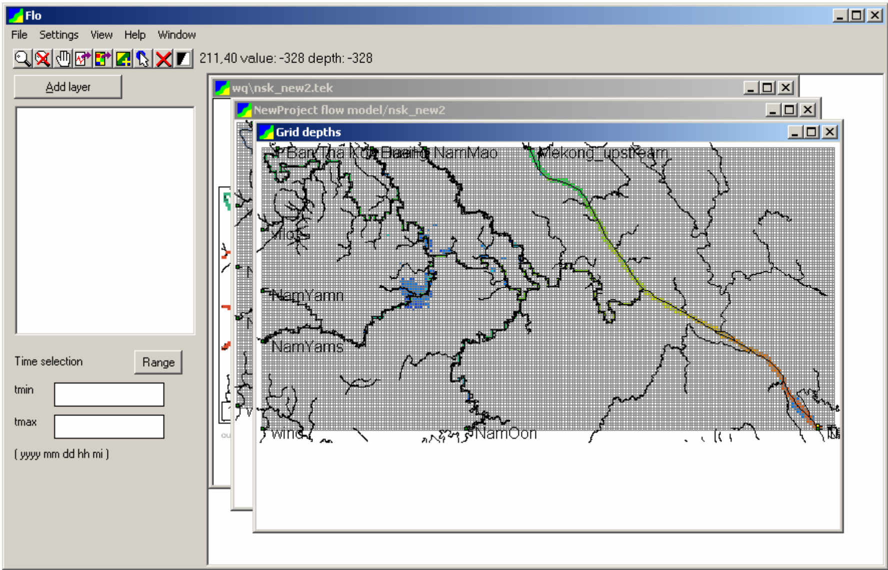
Under progress…? will be available in future under address www.eia.fi

17 WINDOW MANAGEMENT

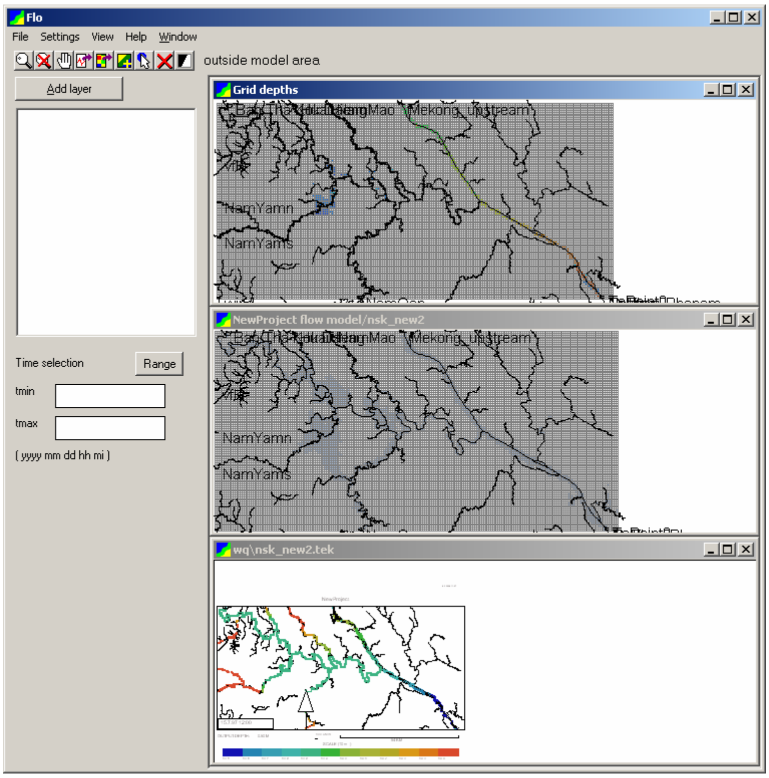
This chapter describes the window management tools.

Under the Window-menu in main menu the user has following window management tools

- Cascade – cascades the open windows



- Tile – tiles the open windows



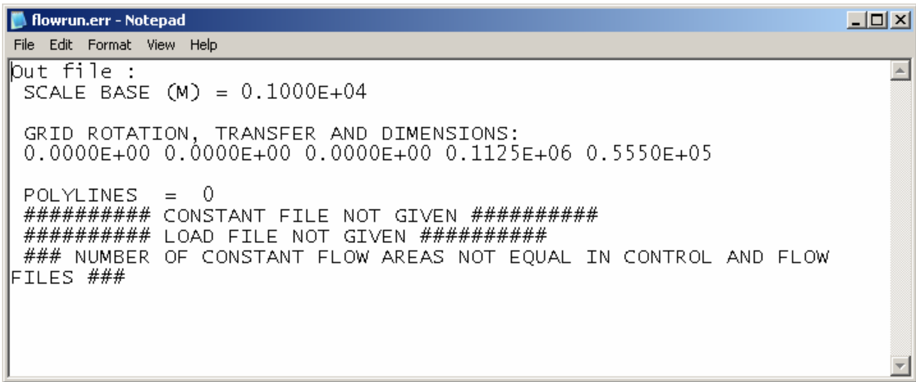
- Close all – closes all the windows
- Active window – shows the list of the open windows and which one is active.

18 TROUBLESHOOTING

There are two basic ways how user interface recognises error situations. The first checking is done when user interface creates input files for the model. Preprocessor does some basic checking, such as existence of input data filse. For instance user prescribed flow data file may have been removed, and the preprocesor would give following error notification:

The point name indicates in what flow data point the missing file has been defined. Because of this it is good practice to give proper names for all control areas and points, unlike in the example the name “vflo”.

The second way for the user interface is to check model out-file. Successful model output results in specific output in the end of the file. If such end is not found user interface presents out-file error message, if present. Observe that messages “CONSTANT FILE NOT GIVEN” and “LOAD FILE NOT GIVEN” are usual warnings and not errors.



Error messages can also be printed in ERROR.ERR-file, which user interface also presents. Numerical errors such as numerical overflow (instability) can output in different ways depending on the compiler, linker and operating system characteristics. For instance “n/a” and “pov overflow” can signify model instability and numerical overflow.

Known issues in the model user interface and model execution are:

- **FLOformula** option using two output exv-files works unreliably in **Field Draw and Compute**.
- **Timeseries Reports**-feature functions only half-automatically and requires user specifications

- If user interface crashes a copy can remain in the memory; this can be killed from Windows Task Manager (right click task bar and select Task Manager, go to Processes and kill viv.exe)
- In some older graphics cards OpenGL graphics doesn't work properly when display hard acceleration is on. For instance model can go into state where it consumes computation time without advancing in actual model simulation. In these casese user should decrease hardware acceleration to basic level or switch off 3D OpenGL graphics in **Model – Animation – Animation options**...

19 EXAMPLES OF MODEL SET-UP

19.1 COMPUTATION OF STATIC FLOW FIELD

A static flow field is computed from an arbitrary initial state (usually no-flow) keeping all the external forcing constant, computing the flow until is converges to a stable pattern. This is typically the simplest computation type since there is no need to set up time-varying external forcing and discharges.

Static flow field computation is useful to check that all the computational options are correctly configured and that the model computation does not get unstable after a long period of computation. Static flow fields can also be used in water quality model.

Typical static flow computations include situations where wind is constant, or there is a constant flow through the model area, or a combination of these. Note that a single constant flow coming into or leaving from the model area cannot be regarded as a static situation, since the amount of water in the model area either decreases or increases.

To compute a static flow field you need to (in addition to setting grid and computation parameters)

- Set a constant wind value using Source data/Wind dialog or
- Set a zero wind value and a constant throughflow through the model area using Source data/Constant flows dialog. Note that the sum of flows leaving and entering the model area should equal to zero
- Compute the model for long enough period (e.g. 100 hours) so that a stable flow pattern emerges

The stability of the flow field can be seen, for example, from surface height or flow speed timeseries. When there is no variation in these timeseries (the line is straight) the flow field is stable. Check timeseries from several locations.

19.2 COMPUTATION OF A DYNAMIC FLOW FIELD

Computation of time-varying flow differs from static flow field computation in such way, that in the static computation the external forcing are constant, whereas in the dynamic computation the forcing vary with time. Dynamic flow computation can be used, for example, to compare computed model results to measured flows.

To compute a dynamic flow you need to (in addition to setting grid and computation parameters)

- Set Time-varying wind data for the period you want to compute using Source data/Wind dialog
- Set Time-varying boundary flows for the period you want to compute using Source data /Constant flows dialog
- Set Model initial state for the start of the computation period using Model/Start- and End fields dialog - or then you can just assume initial no-flow state and start the computation a bit earlier
- Compute the model for the required time period using Model/Run dialog

19.3 Computation of advection of a substance from initial state

Computation of advection of substances from initial state is usually performed using a dynamic flow field. This kind of computation can be used to find out, for example, where to a specific water mass moves to during a given time period. Also the exchange of water between different areas in different conditions can be explored

To compute advection of a substance from initial state you need to

- Set up computation of a dynamic flow field as above
- Set up a computation variable that describes the substance transported in the water using Model/Variables -dialog
- Set up the variable initial values either using Model/Initial values or Model/Variables dialogs.
- Compute the model similarly as in the above dynamic flow field case

19.4 Computation of spreading of a substance from a non-moving source point

Computation of spreading is usually performed using dynamic flow field and initial values for the spreading variables. This kind of computation can be used to follow the spreading of one or more substances from one or more source points.

To compute spreading of substances you need to

- Set up computation of advection of substances from initial state as above
- Set up discharge point locations and discharge amounts using Source data/Discharges dialog
- Compute the model similarly as in the above dynamic flow field case

19.5 Water quality

Set the water quality variable and its parameter values:

- Go to “Model/Variables”
- Either “Edit” or “Add”
- Prescribe variable name (“Code” with 4 capital letters, for instance SEDI)
- Select “Active”
- Don’t select “affects density” other than for “SALI” (salinity) and “TEMP” (temperature)
- select “sediment budget” in case of sediment simulation when needed
- prescribe “Settling coefficient” (cm/d), “Initial value” (mg/l) and “Initial sediment value” (g/m2).
- For statistical output, select “2d-statistics”
- 3 water quality variables can be prescribed. More can be prescribed by changing the model executable.

Set additional parameter values for oxygen (“OXYG”):

- Go to “Model/WQ parameter” and in case of land use data “Model/Land use related parameters” and prescribe parameters.

Select time steps:

- Go to “Model/Time steps”

- Select "Advection" and "Advection multistep". Prescribe maximal advection time step, for instance 3600 s.
- For settling substances select "Settling" and time step.
- For oxygen select "Water quality" and time step.

Set up of boundary values and loads:

- Select "Add item"-button from the control button row (yellow and green button with a +-sign)
- Click either one point or select an area with the mouse and select the control ("Concentration" or "Load")

Set up animation options:

- Go to "Model/Animation/Animation options"
- Select "Variable" from the list.
- Select appropriate scale.

19.6 Floating substances (oil, fish larvae)

Go to "Model/Time steps", check "Particle calculation" and prescribe time step (e.g. 100 s)

Go to "Model/Animation/Advanced animation options" and check "Draw particles" in the "Particle animation" box at the bottom of the dialog window.

Select "Add item"-button from the control button row (yellow and green button with a +-sign), click grid point where spill location is and select "Particle release".

In case of continuous release check "continuous release". Observe that in this case number of particles ("nparticles") can't be large, because in each time step ("dtrelease") "nparticles" are released.

Prescribe water and oil/larvae density.

When particle calculation time step and animation are defined, it is also possible to double click the interactive OpenGL-animation window to release particles.

19.7 Salinity intrusion

Define variable "SALI" (see 5.1).

Select "affects density" (see 5.1).

"Settling coefficient" and "sediment budget" should not be checked.

Define k-e turbulence model (see 3.).

Prescribe boundary salinity values (see 5.1).

Go to "Model/Time steps":

- check "Advection" and "Advection multistep" and give maximal time step (e.g. 1000 s)
- check "Density calculation" and prescribe time step (e.g. 100 – 1000 s)
- check "Diffusion" and give time step (e.g. 10 – 100 s)
- check convective mixing and give time step (e.g. 100 – 1000 s); this mixes water column when heavier water is on top of lighter one.

Go to “Model/Physical parameters/Density and concentration computation” and prescribe “Horizontal diffusion”. In tidal areas diffusion can be high, for instance 600'000 cm2/s.

Select animation parameter to “SALI” (see 5.1).

19.8 Sediment (bed erosion, bed load, sedimentation, bank erosion, mud/ cohesive sediments)

For the definitions including sedimentation follow instructions for water quality.

Go to “Model/Time steps”, check “Erosion” and prescribe time step (e.g. 100 -500 s). Even if flows are read from a dynamic field “Bottom friction” time step needs to be checked.

When impact of waves on erosion is required:

- click “Model/Physical parameters/Fetch”-button and check “dofetch”
- click “Model/Physical parameters/Waves”-button and check “dowave”; select “SPM 1984” (Jonswap is for deep water).
- Click “Model/Physical parameters/Erosion”-button:
- check “doerosion”
- select eroding variable (“SEDI”)
- prescribe critical velocity for erosion (e.g. 10 cm/s); this depends on the quality of the sediment and what kind of flow fields are used: averaged dynamic fields smooth out flow peaks!
- prescribe erosion speed (g/m2/d).

At the moment bank erosion is not explicitly simulated, but work for adding bank stability and erodability in the model is in progress. It is observed that bank erosion seems to correlate strongly on river channel flow, both horizontal and vertical. In order to see vertical velocities go to “Model/Animation/Advanced animation options” and check “Draw vertical velocities”. Prescribe “Vertical scale” in micro meters per second (for instance 400'000 um/s). Vertical velocities are shown either as circles (upward velocity) or crosses (downward velocity). The vertical arrows can be also seen by rotating the animation and seeing the picture sideways.

Bed load and cohesive sediment (mud) simulation has to be turned at the moment in the model code. These are new developments without user interface support.

19.9 Structures, dredging and other structural measures

Blockages such as embankments or roads as well as dredged areas can be accommodated by changing the grid elevations:

Go to “Model/Grid depths”

Select either “Map” (graphical editing) or “Tabular” (table editing)

In “Map” option select with mouse area and prescribe elevations.

When “Settings/Relative depths” in checked in the “Map”-option, the prescribed values will be added to the elevations. This enables for instance easy dredging for a prescribed depth or definition of a road embankment to a prescribed height. Observe that in the model elevations are negative.

Dikes, gates and dike and dam breaks can be also prescribed through (i) “Model/Control structures” and using either graphical or tabular editor, (ii) GIS-files or (iii) specific description files. For the methods and input values see the model manuals, for instance 123D-EIA Model Manual. At the moment graphical editor reads the controls but doesn't show them. This is because the graphics interferes with other user interface functions. The limitation is intended to be corrected in the future model versions.

19.10 Combined 1D/2D/3D simulation

For detailed instructions see 123D-EIA Model Manual. Below is only a rough outline:

Change to RNet 1D model user interface with VivDirSetup.exe programme. Select C:\EIAModels\VIV-VMod folder.

Set up the 1D model to accommodate the river and channel network

Change to 3D model user interface with VivDirSetup.exe programme. Select C:\EIAModels\VIV folder.

Set up the 2D or 3D model that accommodates large rivers, floodplains, lakes, reservoirs and coastal areas (change to C:\EIAModels\VIV interface folder with VivDirSetup.exe programme).

Define structures for both 1D and the 2D/3D models.

Calculate grid combination file with the RNet user interface.

Define specific structures (gates, dikes etc.) that connect the channels to the floodplain.

Run the combined model from the 3D model user interface (model executable has to be combined 123D-executable).

PART II – MODEL EQUATION BACKGROUND

20 BASIC MODEL EQUATIONS

20. BASIC EQUATIONS

20.1 Flow model

First three basic equations which describe the motion of a fluid are introduced:

I continuity equation

$$\frac{D\rho}{Dt} + \rho\mathcal{D} = 0 \tag{20.1}$$

$$\frac{D\rho}{Dt} = \frac{\partial\rho}{\partial t} + \mathbf{u}\cdot\nabla\rho$$

$$\mathcal{D} = \nabla\cdot\mathbf{u},$$

where ρ is the density and $\mathbf{u}$ the velocity vector. $\frac{D}{Dt}$ is the total time derivative or the material derivative and $\mathcal{D}$ the dilation. This equation states the conservation of mass.

II momentum equation

$$\rho\frac{D\mathbf{u}}{Dt} = \rho\mathbf{f} - \nabla p + \nabla\cdot\underline{\boldsymbol{\tau}}, \tag{20.2}$$

or alternatively

$$\frac{\partial^2\rho\mathbf{u}}{\partial t^2} + \nabla\cdot(\rho\mathbf{u}\cdot\mathbf{u}) = \rho\mathbf{f} - \nabla p + \nabla\cdot\underline{\boldsymbol{\tau}}, \tag{20.3}$$

where $\mathbf{f}$ is the volume force per unit mass, p is the pressure and $\underline{\boldsymbol{\tau}}$ is the stress tensor.

III energy equation

$$\rho\frac{De}{Dt} + p\nabla\cdot\mathbf{u} = \Phi - \nabla\cdot\dot{\mathbf{Q}} + S_e \tag{20.4}$$

or

$$\rho\frac{Dh}{Dt} - \frac{Dp}{Dt} = \Phi - \nabla\cdot\dot{\mathbf{Q}} + S_h \tag{20.5}$$

where

e	is the specific internal energy
$h = e + p/\rho$	is the specific enthalpy
$\Phi = \underline{\boldsymbol{\tau}}\cdot\nabla$	is the energy dissipation due to irreversible viscous work
$\dot{\mathbf{Q}} = -k_T\nabla T$	is the heat transfer rate
S_e, S_h	are the source terms
k_T	is the thermal conductivity
T	is the temperature.

The ideal gas equation is

$$p = \rho RT. \tag{20.6}$$

For air $R = 0.287$ kj/kg°K. For ideal gas and for solids and liquids

$$c_p\nabla T = \nabla h, \tag{20.7}$$

where c_p is the constant-pressure specific heat. When c_p is constant (20.7) simplifies into

$$h = c_p(T - T_{ref}). \tag{20.8}$$

This equation can be used to eliminate either temperature of enthalpy in (20.5). A steady state, ideal gas, low-velocity flow with negligible viscous dissipation and constant specific heat would be described by the equation

$$\nabla \cdot (\rho \mathbf{u} T) = \nabla \cdot (\frac{k_T}{c_p} \nabla T) + \frac{S_h}{c_p}. \tag{20.9}$$

When equation (20.2) is solved numerically some kind of time averaging is needed. The time averaging practice starts from the idea that the flow can be represented by

$$u = \overline{u} + u, \tag{20.10}$$

where $\overline{u}$ is the time averaged value and u the turbulent fluctuations. The first problem with this approach is that the the mean flow and fluctuations depend on the time scale of integration. The second problem is that the time averaging process gives rise to additional terms, the so called Reynolds stresses, which can be solved only by more or less intuitive means. A popular approach is to model the turbulent stress in analogous way with the laminar viscosity terms, that is the stresses depends on the velocity differences. E.g. for the x-momentum equations the Reynolds stresses could be described by

$$\begin{aligned} -\overline{\rho u u} &= \rho \nu_{tu} \frac{\partial u}{\partial x} \\ -\overline{\rho u v} &= \rho \nu_{tu} \frac{\partial u}{\partial y} \\ -\overline{\rho u w} &= \rho \nu_{tu} \frac{\partial u}{\partial z}. \end{aligned} \tag{20.11}$$

The difficult task is to determine the turbulent eddy viscosity (or diffusivity) ν_{tu}. One has to utilize more or less ad hoc methods. One can use constant values, introduce some simple density, scale etc. dependent expressions or apply sophisticated turbulence models such as the k-ε–model (k is the dissipation of the turbulent energy and ε the turbulent kinetic energy). The expression for the turbulent diffusion (or mixing) coefficient can also be derived by dimensional analysis, see [20, pp. 17 – 19 and p. 50]. According to this analysis diffusion coefficient depends on the mixing length L

$$\nu_{tu} = K_\nu L^{\frac{4}{3}} \tag{20.12}$$

where K_ν is a constant. According to the measurements in natural water bodies the exponent in (20.12) is slightly smaller than 4/3, sometimes even 1.

When the liquid is incompressible the partial time derivative of the density can be set to zero. Often it can also be assumed that the density variability is appreciably smaller than velocity variability, $|\mathbf{u} \cdot \nabla \rho| \ll |\rho \nabla \cdot \mathbf{u}|$ (the Boussinesq-approximation). Then the equation (20.2) reduces to

$$\nabla \cdot \mathbf{u} = 0. \tag{20.13}$$

When hydrostatic assumption (in the equation for the vertical velocity component other terms than the gravitational force and pressure are negligible) is valid the equation of the vertical velocity component reduces to

$$\frac{\partial p}{\partial z} = -g\rho. \tag{20.14}$$

The force-term in the horizontal momentum equations usually contains only the Coriolis-force. It is also usually assumed for the estuarine and marine flows that in the horizontal momentum equations a constant density ρ_0 can be substituted for the density ρ. ρ_0 is most appropriately chosen to be the average density.

The validity of above mentioned assumptions can be tested with a order of magnitude analysis. For most estuarine and marine flows these assumptions are well justified ,see e.g. [20, pp. 15 – 16].

After introducing the simplifications mentioned above the equations used in the current 3D marine and estuarine flow model can be summarized:

$$\frac{\partial u}{\partial t} = fv - \frac{1}{\rho_\circ}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(\nu_{\text{hor}}\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(\nu_{\text{hor}}\frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(\nu_{\text{ver}}\frac{\partial u}{\partial z}) - \mathbf{u} \cdot \nabla u \tag{20.15}$$

$$\frac{\partial v}{\partial t} = -fu - \frac{1}{\rho_\circ}\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(\nu_{\text{hor}}\frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(\nu_{\text{hor}}\frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(\nu_{\text{ver}}\frac{\partial v}{\partial z}) - \mathbf{u}\cdot\nabla v \quad (20.16)$$

$$\frac{\partial p}{\partial z} = -g\rho \quad (20.17)$$

$$\nabla\cdot\mathbf{u} = 0 \quad (20.18)$$

$\mathbf{u}$	is the velocity vector
u, v, w	are the x, y and z velocity components (m/s)
t	is the time (s)
p	is the pressure (Pa)
$\rho_\circ$	is the reference (average) density (kg/m^3)
f	is the Coriolis coefficient
g	is the acceleration of the earths gravity (m/s^2)
$\nu_{\text{hor}}, \nu_{\text{ver}}$	are the horizontal and vertical turbulent momentum diffusion coefficients (m^2/s).

In the above mentioned equations the momentum advection, $(\mathbf{u}\cdot\nabla)\mathbf{u}$, can usually be ignored. The cases when it cant be ignored are for instance high velocity jets. The horizontal and vertical momentum diffusion coefficients are indicated separately to stress the usual order of magnitude difference between their values. Pressure p can be divided into the barotropic (outer) and baroclinic (inner) parts:

$$\begin{aligned} p &= p_e + p_i \quad (20.19) \\ p_e &= g\zeta\rho_0 + p_a \\ p_i &= g\int_z^\zeta \Delta\rho\, dz, \end{aligned}$$

where ζ is the surface elevation, p_a the air pressure and $\Delta\rho = \rho - \rho_0$ the density difference. Quadratic boundary conditions are usually assumed on the surface:

$$\begin{aligned} \tau_{sx} &= K_s\rho_a u_a\sqrt{u_a^2 + v_a^2} \\ \tau_{sy} &= K_s\rho_a v_a\sqrt{u_a^2 + v_a^2}, \end{aligned} \quad (20.20)$$

where τ_{sx} and τ_{sy} are the surface wind stress components, K_s is the wind friction coefficient, ρ_a is the air density and u_a and v_a are the wind velocity components. The wind stress is often the most important forcing factor in water circulation and thus the accurate modelling of it is of primary importance. First of all the wind velocity $\mathbf{u_a}$ should be accurate in every point. This would require extensive measurements and a wind model because the wind may vary considerably from one location to another. Also the friction coefficient K_s should be modeled accurately. For instance surface roughness (short waves and ripples) affect the friction coefficient. Often modelling is done however without proper knowledge of the real wind velocities over the water body. Then the wind friction must be correlated with wind measurements on some nearby locale (usually a weather station or an airport) through the wind friction coefficient. In this case the wind friction coefficient should take into account the wind measurement height, surface roughness (different wind velocities e.g. for water, field, forest), the distance the wind has traveled from the shore (wind fetch), the angle the wind turns on the boundary between land and water etc. In Finland Juha Sarkkula has studied the wind stress in natural basins based on extensive current and wind measurements and modelling applications [9].

Bottom stress formulation has been either linear or quadratic. The quadratic form is

$$\begin{aligned} \tau_{bx} &= K_{bq}\rho_0 u\sqrt{u^2 + v^2} \\ \tau_{by} &= K_{bq}\rho_0 v\sqrt{u^2 + v^2} \end{aligned} \quad (20.21)$$

and the linear one

$$\begin{aligned} \tau_{bx} &= K_{bl}u \\ \tau_{by} &= K_{bl}v. \end{aligned} \quad (20.22)$$

The linear bottom friction coefficient can be a constant or a depth dependent function e.g.

$$K_{bl} = \frac{r}{H} \quad (20.23)$$

or

$$K_{bl} = rH, \tag{20.24}$$

where H is the depth. Bottom friction depends also on the bottom inclination and turbulence, see [7, pp. 74 – 75] for the formulations. According to the Delft Hydraulics Laboratory the usual approximation should be replaced with a more accurate one where the profile of the stress is approximated near the bottom, see [7, pp. 528 – 530].

The principal tool chosen for the Bothnian Bay study was the 3D model introduced by T.J. Simons [15], [16], [12]. The simple idea behind the model is to divide the calculation of the velocities into two parts. First part (vertically integrated currents) contains the free surface elevation calculation. The second part (velocity differences) doesnt contain the free surface equations and is computationally much more cheaper. The model is explicit, which implies easy programmability and correct time behavior. The model was chosen mainly because of the possibility to change the code easily and because of the very reasonable computer costs required by the model. The computer costs were main concern because of the large time spans of the Bothnian Bay simulations. The optimal computer costs of the model were verified by comparing the model computer time consumption with that of two 2D models (explicit Hansen model (VTT) and one implicit model (VITUKI)) and a 3D model (Phoenix).

20.2 Temperature model

Mathematically the concentration changes in the water bodies are described by the equation

$$\frac{\partial c}{\partial t} = -\mathbf{u}\cdot\nabla c + \nabla\cdot(D_{hor}\nabla c) + \frac{\partial}{\partial z}(D_{ver}\frac{\partial c}{\partial z}) + L_c + S_c, \tag{20.25}$$

where c is the concentration, D_{hor} horizontal and D_{ver} vertical eddy diffusion coefficient, L_c biological-chemical-physical processes and S_c contains the sinks and sources. This equation is applicable for temperature, salinity and the various water quality factors.

For the temperature modelling the important factor is L_c in equation (**??**). This will be studied in the next sections.

The mathematical model for the temperature modelling will be developed now. The principle of energy conservation leads into the following equation

$$\underbrace{\tfrac{\partial}{\partial t}(\rho T)}_{\text{local change}} + \underbrace{\tfrac{\partial}{\partial x_j}(u_j\rho T)}_{\text{transport}} = \underbrace{-\tfrac{\partial}{\partial x_j}(\rho\overline{u_j T})}_{\text{turbulent diffusion}} + \underbrace{\tfrac{\partial}{\partial x_j}(\tfrac{\mu_l}{\rho\sigma_l}\tfrac{\partial}{\partial x_j}(\rho T))}_{\text{molecular diffusion}} + \underbrace{\tfrac{S_H}{c}}_{\text{source and sink term}} \tag{20.26}$$

T	is water temperature (°C)
ρ	is density of water ($\frac{\mathrm{kg}}{\mathrm{m}^3}$)
c	is specific heat of water = 4.2103 ($\frac{\mathrm{J}}{^\circ\mathrm{C}\cdot\mathrm{kg}}$)
μ_l	is laminar viscosity ($\frac{\mathrm{kg}}{\mathrm{m}}\cdot\mathrm{s}$)
σ_l	is laminar Prandtl number
u_j	is velocity ($\frac{\mathrm{m}}{\mathrm{s}}$)
x_j	is space coordinate (m)
S_H	is source and sink term ($\frac{\mathrm{J}}{\mathrm{m}^3\cdot\mathrm{s}}$).

Turbulent heat transfer is modelled analogous to the laminar viscosity:

$$\rho\overline{u_j T} = \frac{\partial\mu_t}{\partial\sigma_t}\frac{\partial T}{\partial x_j} \tag{20.27}$$

where σ_t is the turbulent Prandtl number. Laminar viscosity in environmental flows is insignificant compared to the turbulent viscosity. When water density is assumed to be constant equation (20.27) is simplified into

$$\frac{\partial T}{\partial t} + u_j\frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j}(\frac{\nu_t}{\sigma_t}\frac{\partial}{\partial x_j}(T)) + \frac{S_H}{\rho c} \tag{20.28}$$

where ν_t (m/s) is kinematic eddy viscosity. See [17] for detailed derivation of μ_t and σ_t.

Tab. 20.1: *a and b in equation (20.32) as functions of high clouds [1].*

cloudiness	0 - 5%	5 - 55%	55 - 95%	95 -100%
a	1.18	2.20	0.95	0.33
b	-0.77	-0.97	-0.75	-0.45

20.2.1 Source and sink term

The source and sink term S_H consists of internal absorption of solar radiation and heat discharges/sinks (cooling waters, heat pumps). The infrared part of solar and atmospheric radiation is absorbed within 1 m from surface so that it is taken into account by the boundary values. The incoming shortwave radiation R_{SI} (W/m^2) is absorbed exponentially:

$$S_H = \frac{\partial}{\partial z}(R_{SI}e^{\beta z}) = \beta R_{SI}e^{\beta z} \tag{20.29}$$

where β (1/m) is extinction coefficient and z vertical coordinate. β depends strongly on the clarity of the water and can be estimated by secchi disc value d_S (m) [13]

$$\beta = 1.65/d_S. \tag{20.30}$$

The incoming shortwave radiation R_{SI} depends on the reflectivity of the water surface [4]:

$$\begin{aligned}
R_{SI} &= (1 - r_s)R_S \qquad (20.31)\\
r_s &= a\alpha_a^b \\
\alpha_a &= \arcsin(\sin\phi\sin\sigma_d + \cos\phi\cos\sigma_d\cos h_r) \\
\sigma_d &= 23.45\frac{\pi}{180}\cos(\frac{2\pi}{365}(172 - D)) \\
h_r &= \frac{\pi}{24}(h_s + \frac{4}{60}\psi \pm 12) \\
a &= 2.20 + \frac{C_r^{0.7}}{4.0} - \frac{(C_r^{0.7} - 0.4)^2}{0.16} \\
b &= -1.02 + \frac{C_r^{0.7}}{16.0} + \frac{(C_r^{0.7} - 0.4)^2}{0.64} \\
C_r &= 1 - R_S/R_{Smax}
\end{aligned}$$

R_S	is shortwave radiation reaching the water surface (W/m)
r_S	is reflectivity of the water surface
α_a	is solar altitude (rad)
ϕ	is geographic latitude (rad)
h_r	is local hour angle of the sun (rad)
h_s	is standard time (h)
ψ	is distance between the meridian of the observer and the standard meridian of the time zone (deg); east longitude $\Rightarrow -$, west longitude $\Rightarrow +$
σ_d	is declination of the sun (rad)
D	is number of the day (1 – 365)
R_{Smax}	is theoretical maximum solar shortwave radiation.

In the equation for h +-sign before 12 is taken before noon and −-sign after noon. Constants a and b depend actually on the cloud cover [1], but the expression in question has the advantage that no cloud information is needed when shortwave radiation is measured. Table 20.1 gives the averaged values of a and b for different cloud covers.

R_{Smax} can be computed by an empirical formula [10], [11]:

$$\begin{aligned}
R_{Smax} &= R_{Se}(0.99 - 0.17m_\circ) \qquad (20.32)\\
R_{Se} &= I_\circ \sin\alpha_a \\
m_\circ &= 1/[\sin\alpha_a + 0.15(\alpha_a + 3.885)^{-1.253}]
\end{aligned}$$

R_{Se}	is extraterrestrial radiation
$I_\circ$	is the solar constant $14.0 \cdot 10^2$ W/m
α_a	is solar altitude (rad)
$m_\circ$	is optical air mass.

When total radiation values are available R_S can be approximated by

$$R_S = r_s R_t \tag{20.33}$$

R_t	is total radiation reaching the ground
r_s	≈ 0.6 [6].

R_S can be also calculated from total radiation and amount of longwave radiation (section 20.2.2). Widely used formula is due to Kennedy [10]:

$$R_S = R_{Se}(1 - 0.65C_l^2) \tag{20.34}$$

R_{Se}	is extraterrestrial radiation
C_l	is cloudiness (0 - 1).

20.2.2 Boundary values

Boundary values for temperature calculation consist of heat fluxes into the surface water, bottom and banks. Shortwave radiation was discussed on the section 20.2.1 as it affects the whole water body. Atmospheric longwave radiation is absorbed within the first meter from the surface and can be thus modeled as a heat flux through the surface. According to the Stefan-Bolzman law longwave radiation is

$$R_L = \varepsilon_a \sigma_B (T_a + 273)^4 \tag{20.35}$$

R_L	is amount of longwave radiation (W/m^2)
ε_a	is atmospheric emissivity
σ_B	is Stefan-Bolzman constant $= 5.67 \cdot 10^{-8} \frac{\mathrm{W}}{\mathrm{m}^2 {}^\circ\mathrm{K}^4}$
T_a	is air temperature (°C), usually measured 2 m above the water surface.

Various empirical relations have been developed for ε_a. Generally ε_a is expressed as a function of the air temperature and/or humidity. The frequently used Swinbank formula doesnt depend on the local temperature-humidity regime of the locale [18],[8]:

$$\varepsilon_a = 0.93710^{-5}(T_a + 273)^2. \tag{20.36}$$

Brutsaert has proposed theoretical vapor pressure dependent equation which is in good agreement with empirical formulas [5]:

$$\begin{aligned} \varepsilon_a &= 0.553 e_a^{1/7} \qquad (20.37) \\ e_a &= H_r e_{as} \\ e_{as} &= 6.108 \exp\left(\frac{17.27 T_a}{T_a + 237.44}\right) \quad [19] \end{aligned}$$

e_a	is the vapor pressure of the air at the temperature T_a (mb)
H_r	is relative humidity (0 - 1)
e_{as}	is saturation vapor pressure (mb)
T_a	is air temperature (°C).

Bolz-formula is often used to take into account the effect of clouds on longwave radiation [3]:

$$\varepsilon_a = \varepsilon_c(1 + k_a C_l^2) \tag{20.38}$$

ε_c	is atmospheric emissivity under clear skies
C_l	is cloudiness (0 - 1)
k_a	is factor which depends upon the type and height of the clouds and solar altitude; a mean value of 0.17 can be used.

The reflectivity of the water surface for longwave radiation is 0.030 so the incoming longwave radiation is given by

$$R_{LI} = 0.970 R_L \tag{20.39}$$

The longwave radiation from water body has the largest magnitude among all the heat exchange components. It is given by Stefan-Bolzman law:

$$R_{LO} = \varepsilon_w \sigma_B (T_w + 273)^4 \tag{20.40}$$

R_{LO}	is amount of outgoing longwave radiation (W/m^2)
ε_w	is emissivity of the water surface = 0.970
σ_B	is Stefan-Bolzman constant
T_w	is water surface temperature (°C).

The evaporative heat transfer can be quite considerable. Evaporation consumes energy according to the following equation:

$$\begin{aligned} Q_e &= \rho L E \qquad (20.41) \\ L &= 25 \cdot 10^5 - 2390 T_w \\ E &= (a_c + b_c W)(e_w - e_a) \\ e_w &= 6.108 \exp\left(\frac{17.27 T_w}{T_w + 237.44}\right) \end{aligned}$$

Q_e	is evaporative heat flux (W/m)
ρ	is density of water (kg/m3)
L	is the latent heat of vaporization (J/kg)
E	is the volumetric evaporation rate per unit area of the water surface (m/s)
T_w	is water surface temperature (°C)
a_c	is the free convection factor; a_c depends on virtual temperature gradient; often $a \approx 0$
$b_c W$	is the forced convection factor where W is wind speed (m/s); b_c depends eg. on the wind fetch, but here b_c $1.13 \cdot 10^{-9}$/mb [14] when W and e_a are measured 2 m above the water surface; this value is recommended by TVA [2]
e_a	is the vapor pressure of air at the temperature T_a (mb)
e_w	is surface vapor pressure (mb).

The heat flux carried by the evaporated mass of water is insignificant. Heat conduction through surface is regularly given by

$$\begin{aligned} Q_{ca} &= r_B Q_e \qquad (20.42) \\ r_B &= c_c \frac{p_a}{1000} \left(\frac{T_w - T_a}{e_s - e_a}\right) \end{aligned}$$

Q_{ca}	is heat conduction through water surface (W/m)
r_B	is the Bowens ratio
p_a	is air pressure (mb)
c_c	is constant = 0.61/°C .

Heat conduction through bottom can be given as

$$Q_{cb} = -K_t \frac{\partial T}{\partial z} \tag{20.43}$$

where K_t is thermal conductivity between water and bottom ($\frac{\mathrm{W}}{\mathrm{m^\circ C}}$).

20.3 Water quality modelling

$$
\begin{aligned}
P_{use} &= K_{Pa} + K_{Pb}C_P - K_{Psto}(C_{mor} + C_A) \\
N_{use} &= K_{Na} + K_{Nb}C_N - K_{Nsto}(C_{mor} + C_A) \\
R_{gr} &= K_{Agr1}\left(\frac{P_{use}}{P_{hsat} + P_{use}}\right)\left(\frac{N_{use}}{N_{hsat} + N_{use}}\right)\left(\frac{L}{L_{hsat} + L}\right)C_A \\
R_{resp} &= \left(K_{res1} + K_{res2}C_A\right)C_A \\
SED_x &= \frac{\partial\left(\left(K_{sed1_x} + K_{sed2_x}C_x\right)C_x\right)}{\partial z} \qquad (20.44) \\
\frac{\partial C_{mor}}{\partial t} &= -SED_{mor} + R_{resp} \\
\frac{\partial C_P}{\partial t} &= -SED_P - SED_{mor}K_{Psto} \\
\frac{\partial C_N}{\partial t} &= -SED_N - SED_{mor}K_{Nsto} \\
\frac{\partial C_A}{\partial t} &= -SED_A + R_{gr} - R_{resp}
\end{aligned}
$$

where:

C_{mor}	is the dead algae concentration (mg l^{-1})
C_P	is the phosphorus concentration (μg l^{-1})
C_N	is the nitrogen concentration (μg l^{-1})
C_A	is the live algae concentration (mg l^{-1})
R_{gr}	is the algal growth rate (mg l^{-1} d^{-1})
R_{resp}	is the algal respiration (mortality) rate (mg l^{-1} d^{-1})
P_{use}	is the phosphorus fraction that can be utilized by the algae (μg l^{-1})
N_{use}	is the nitrogen fraction that can be utilized by the algae (μg l^{-1})
K_{Psto}	is the phosphorus stochiometric coefficient (μg P/ mg A)
K_{Pa}	is the phosphorus utilization constant (μg l^{-1})
K_{Pb}	is the phosphorus utilization coefficient
K_{Agr1}	is the maximal algal growth rate (d^{-1})
P_{hsat}	is the phosphorus half saturation constant (μg l^{-1})
L	is the incoming shortwave radiation (W m^{-2} d^{-1})
K_{res1}	is the linear respiration (mortality) coefficient (d^{-1})
K_{res2}	is the quadratic respiration (mortality) coefficient ($d^{-1}mg^{-1}$)
SED_x	is the sedimentation rate for component x (μg or mg d^{-1})
K_{sed1_x}	is the linear sedimentation coefficient (cm d^{-1})
K_{sed2_x}	is the quadratic sedimentation coefficient (cm d^{-1} / μg or mg l^{-1})

References

[1] E.A. Anderson. Energy budget studies, water-loss investigations. Technical report, USGS Professional Paper 269, Lake Hefner studies, 1954.

[2] Tennesee Valley Authority. Heat and mass transfer between a water surface and the atmosphere. Report 14, Water Resources Research Laboratory, 1970.

[3] S.H. Bolz. The dependendence of the infrared counter-radiation on cloud mass (in german). *z.f. Meteorologie*, 3(5–6), 1949.

[4] D.K. Brady, W.L. Graves, and J.C. Geyer. Surface heat exchange at power plant cooling lakes. Publ. 69-901, Edison Electric Institute, New York, 1969.

[5] W. Brutsaert. On a derivable formula for long-wave radiation from clear skies. *Water Resources Research*, 11(5):742.

[6] J.M.K. Dake and D.R.F. Harleman. Thermal stratification in lakes: analytical and laboratory studies. *Water Resources Research*, 5(2):p. 484, 1969.

[7] Hugo B. Fischer, editor. *Transport Models for Inland and Coastal Waters, Proceedings of a Symposium on Predictive Ability*. Academic Press, 1981. ISBN 0-12-258152-0, 542 pp.

[8] S.R. Idso and R.D. Jackson. Thermal radiation from atmosphere. *J. of Geophysical Research*, 74, Oct. 1969.

[9] J. Józsa, J. Sarkkula, and R. Tamsaly. Calibration of modelled shallow lake flow using sind field modification. Technical report, VITUKI, National Board of Waters and Environment and Estonian Academy of Sciences, 1990. 6 pp.

[10] R.E. Kennedy. Computation of daily insolation energy. *Bull. of the American Meteorological Society*, 30, 1944.

[11] W.H. Klein. Calculation of solar radiation and the solar heat load on man. *J. of Meteorology*, 5(4), 1948.

[12] J. Koponen. A three dimensional current and water quality model. Master's thesis, Helsinki University of Technology, 1984. 98 pp.

[13] K. Lehtinen. *Mathematical modelling of temperature in a water body (in Finnish)*. Publ. of National Board of Waters. National Board of Waters in Finnland, Helsinki, 1984. p. 21.

[14] J.J. Marciano and G.E. Harbeck. Mass-transfer studies. Technical report, USGS Professional Paper 2657, Lake Hefner studies, 1954.

[15] T.J. Simons. Circulation models for lakes and inland seas. *Can. Bull. Fisheries and Aquatic Sci.*, (203):1–145, 1980.

[16] T.J. Simons and Kielman J. Some aspects of baroclinic circulation models. In K. Hunter, editor, *Hydrodynamics of Lakes*, number 286 in CISM Courses and Lectures, pages 235–285. Springer-Verlag, Wien–New York, 1984.

[17] U. Svensson. A mathematical model of the seasonal thermocline. Technical Report 1002, University of Lund, Dept. of Water Resources Eng., Sweden, 1978. pp. 111–150.

[18] W.C. Swinbank. Long-wave radiation from clear skies. *Quarterly J. of Royal Meteorological Society*, 89, 1963.

[19] O. Tetons. About some meteorological concepts (in german). *z. Geophysik*, 6, 1930.

[20] Markku Virtanen. Description of flow and mixing in water bodies with a 2-dimensional mathematical model (in finnish). Technical licenciate thesis, Helsinki University of Technology, 1977. 128 pp.

21 MATHEMATICAL DESCRIPTION OF WATER FLOW

21.1 Fundamental Equations

The motion of a fluid particle on the surface of the earth is governed by the Navier-Stokes equation of motion (the force balance equation) /1/

$$\tilde{\rho}\frac{d\tilde{\vec{v}}}{dt} = \tilde{\rho}\frac{\partial \tilde{\vec{v}}}{\partial t} + \tilde{\rho}\tilde{\vec{v}} \circ \nabla \tilde{\vec{v}} = -\nabla \tilde{p} + \tilde{\rho}\vec{g} \circ \vec{\vec{I}} - 2\tilde{\rho}\vec{\omega} \times \tilde{\vec{v}} + \tilde{\rho}\upsilon^{m}\nabla^{2}\tilde{\vec{v}} \quad (1)$$

Where

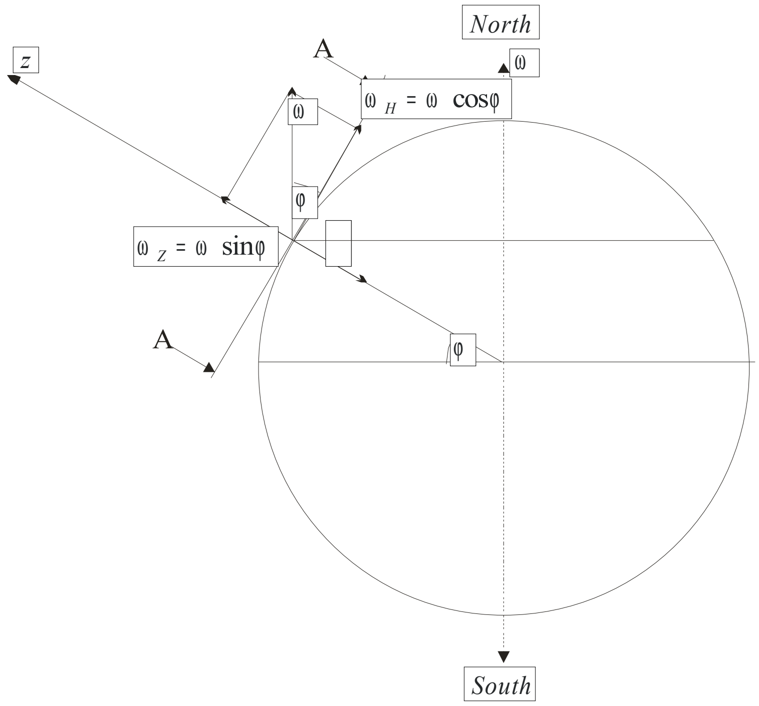
$\tilde{\vec{v}}$	momentaneous flow velocity vector (m s^{-1})
$\tilde{\rho}$	momentaneous density of water (kg m^{-3})
$\tilde{p}$	momentaneous pressure (N m^{-2})
$\vec{g}$	gravity acceleration vector (m s^{-2})
$\vec{\vec{I}}$	unit matrix of the co-ordinate system (-)
$\vec{\omega}$	angular velocity vector of earth's rotation (s^{-1})
υ^{m}	kinematic, molecular viscosity of the water (m^{2} s^{-1})
t	time (s)
∇	gradient operator (grad) (m^{-1})
$\nabla \cdot$	divergence operator (div) (m^{-1})
∇^{2}	Laplace operator (div grad) (m^{-2})

The forces included are (from left to right) local acceleration, advective (or convective) accelerations, pressure gradient, gravitation, Coriolis force and molecular viscosity.

In addition to this there is the continuity equation (mass conservation equation) /3/

$$\frac{\partial \tilde{\rho}}{\partial t} + \nabla \cdot \left(\tilde{\rho}\tilde{\vec{v}}\right) = 0 \quad (2)$$

Globe of the earth:



A-A, x-y-plane on the surface of the Earth:

Figure 19. Coordinate system used in the EIA 3D model.

21.2 Approximation of 3-dimensional Equations

By fixing a coordinate system on the surface of the earth with z pointing upwards (Figure 19), the Navier-Stokes equation can be written out explicitly in component form. By comparing the order of magnitude of different terms in the z-component equation, it can be decided that all other terms are negligible compared to the pressure gradient and gravitation. Thus, using this hydrostatic assumption /2/, the z-component of the Navier-Stokes (Equation 2) can be written as

$$\frac{\partial \tilde{p}}{\partial z} = -\tilde{\rho} g \tag{3}$$

Due to the uncompressibility of water the density differences are much smaller than the velocity differences. Thus, according to this uncompressibility assumption (Boussinesq-approximation /3/), the density variations can be neglected elsewhere except in the buoyancy term, and the continuity equation gets the form

$$\nabla \cdot \tilde{\vec{v}} = 0 \tag{4}$$

By dividing the momentaneous values $\tilde{\vec{v}}, \tilde{p}, \tilde{\rho}$ to time averages $\vec{v}, p, \rho$ and turbulent fluctuations $\vec{v}', p', \rho'$, the equations (1), (3) and(4) can be integrated over the observational time period T, which must be supposed to be long compared to the time scale of turbulent (statistical) fluctuations τ, but short compared to the scale of the dynamical (deterministic) behaviour Δt of the flow. By introducing the concept of turbulent viscosity, i.e. by assuming the time average of the quadratic fluctuations proportional to the gradient of the average velocity /2, 3/:

$$\frac{1}{T}\int_{o}^{T} v'_{\alpha}\, v'_{\beta}\, dt = \upsilon^{t}_{\alpha,\beta}\,\frac{\partial v_{\beta}}{\partial \alpha}, \quad \alpha = x, y, z \quad \beta = x, y, z \tag{5}$$

the fluctuations can be eliminated and a closed system of equations acquired. Taking into account the component expansions of the vector and matrix terms (Figure 19), the following equations result in

$$\begin{aligned}&\frac{\partial v_X}{\partial t} + v_X\frac{\partial v_X}{\partial x} + v_Y\frac{\partial v_X}{\partial y} + v_Z\frac{\partial v_X}{\partial z} = \\ &= -\frac{1}{\rho}\frac{\partial p}{\partial x} - 2\omega\left(v_Z\cos\phi\cos\theta - v_Y\sin\phi\right) + \\ &+ \frac{\partial}{\partial x}\left[\left(\upsilon^{t}_{xx} + \upsilon^{m}\right)\frac{\partial v_X}{\partial x}\right] + \frac{\partial}{\partial y}\left[\left(\upsilon^{t}_{xy} + \upsilon^{m}\right)\frac{\partial v_X}{\partial y}\right] + \\ &+ \frac{\partial}{\partial z}\left[\left(\upsilon^{t}_{xz} + \upsilon^{m}\right)\frac{\partial v_X}{\partial z}\right]\end{aligned} \tag{6}$$

$$\begin{aligned}&\frac{\partial v_y}{\partial t} + v_x\frac{\partial v_y}{\partial x} + v_Y\frac{\partial v_y}{\partial y} + v_z\frac{\partial v_y}{\partial z} = \\ &= -\frac{1}{\rho}\frac{\partial p}{\partial y} - 2\omega\left(v_x\sin\phi - v_z\cos\phi\cos\theta\right) + \\ &+ \frac{\partial}{\partial x}\left[\left(\upsilon^{t}_{yx} + \upsilon^{m}\right)\frac{\partial v_y}{\partial x}\right] + \frac{\partial}{\partial y}\left[\left(\upsilon^{t}_{yy} + \upsilon^{m}\right)\frac{\partial v_Y}{\partial y}\right] + \\ &+ \frac{\partial}{\partial z}\left[\left(\upsilon^{t}_{yz} + \upsilon^{m}\right)\frac{\partial v_Y}{\partial z}\right]\end{aligned} \tag{7}$$

$$\frac{\partial p}{\partial z} = -\rho g \tag{8}$$

$$\frac{\partial v_X}{\partial x} + \frac{\partial v_Y}{\partial y} + \frac{\partial v_z}{\partial z} = 0 \tag{9}$$

where

v_x, v_y, v_z are the components of the time-average flow velocity (m/s)

ρ is the time-averaged density of the water (kg/m^3)

p the time-averaged pressure (N/m^2)

ω the magnitude of the earth's angular velocity (s^{-1})

ϕ the geographical latitude (rad)

θ the angle between x-axis and the East-direction (rad)

$\vec{\upsilon}^{t}_{x} = \left(\upsilon^{t}_{xx}, \upsilon^{t}_{xy}, \upsilon^{t}_{xz}\right)$ defined by (5) (m^2/s)

$\vec{\upsilon}^{t}_{y}$ similar y-component (row) vector of the turbulent viscosity matrix $\vec{\vec{\upsilon}}^{t}$ (m^2/s)

The 3-dimensional flow field can be solved from the above equations (6…9) together with the boundary conditions

$$p = p_a(x, y) \quad \text{on the surface of water } (z = \eta) \tag{10}$$

$$\upsilon^{t}_{z} \frac{\partial \vec{V}}{\partial z} = \vec{\tau}/\rho \quad \text{on the surface of water } (z = \eta) \tag{11}$$

$$-\upsilon^{t}_{z} \frac{\partial \vec{V}}{\partial z} = \vec{\tau}_B/\rho \quad \text{on the bottom } (z = -h) \tag{12}$$

$$\vec{V} = \vec{V}_{river} \quad \text{at possible river inflows and out flows} \tag{13}$$

where

$p_a(x, y)$ = atmospheric pressure on the surface of water (N/m^2)

$\vec{\tau}$ = wind shear stress vector on the surface of water (N/m^2)

$\vec{\tau}_B$ = shear stress vector due to bottom friction (N/m^2)

$\vec{V}_{river}$ = flow velocities at the river cross-sections (N/m^2)

$\left(\nu^{t}_{z} = \nu^{t}_{xz} = \nu^{t}_{yz}\right)$

21.3 Depth Integration

From hydrostatic equation (3) with the boundary condition (10) the pressure p can be solved.

$$p(x, y, z) = p_a(x, y) + g \int_{z}^{\eta} \rho\,(x, y\xi)\,d\xi \tag{14}$$

and substituted into (6) and (7). By dividing the observational time averages $\vec{V}$, p, ρ, into depth averages $\vec{u}$, $\vec{p}$, $\vec{\rho}$, $\vec{\vec{V}}^{t}$ and vertical fluctuations $\hat{\vec{V}}$, $\hat{p}$, $\hat{\rho}$, $\hat{\vec{\vec{\upsilon}}}$; and by taking into account the boundary conditions (11), (12); a depth integration from bottom (z = -h) to the surface (z = η) can be carried out, similarly to the previous time averaging. Incorporating the assumptions of unstratification, small vertical velocities and their integrals compared to the horizontal velocities and to the total depth, slow rate of bottom variation compared to the surface fluctuations, and finally the concept of shear viscosities similar to the turbulent ones (Eq. 5); this yields the following equations

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial}{\partial x}\left(\frac{P_a}{\bar{\rho}} + g\eta\right) + fv + \frac{\tau_x + \tau_{Bx}}{H\bar{\rho}} + \\ + \frac{\partial}{\partial x}\left(\upsilon_{xx}\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon_{xy}\frac{\partial u}{\partial y}\right) \tag{15}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial}{\partial y}\left(\frac{P_a}{\bar{\rho}} + g\eta\right) - fu + \frac{\tau_y + \tau_{By}}{H\bar{\rho}} + \\ + \frac{\partial}{\partial x}\left(\upsilon_{yx}\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon_{yy}\frac{\partial v}{\partial y}\right) \tag{16}$$

$$\frac{\partial \eta}{\partial t} + \frac{\partial (H u)}{\partial x} + \frac{\partial (H v)}{\partial y} = 0 \tag{17}$$

where

u,v	are the depth-averaged mean velocity components (m/s)
f	$= 2\omega \sin\phi$, the coriolis coefficient
η	the surface elevation (m)
H	$= h + \eta$, the total depth of the water column (m)
τ_x, τ_y	the x and y components of the wind shear (N/m^2)
τ_{Bx}, τ_{By}	the x and y components of the bottom shear (N/m^2)
$\upsilon_{xx}, \upsilon_{xy}, \upsilon_{yx}, \upsilon_{yy}$	the elements of the combined eddy viscosity matrix

$$\vec{\upsilon} = \vec{\upsilon}^{\,t} + \vec{\upsilon}^{\,m} + \vec{\upsilon}^{\,s} \qquad (m^2/s)$$

Concerning the accuracy of the approximations both hydrostatic and uncompressibility assumptions are physically well justified and in praxis very accurate $(o(10^{-6})$ and $o(10^{-3})$ respectively; the same applies to the neglection of bottom fluctuations $(o(10^{-6})$ and vertical velocities $o(10^{-2})$ as well. In time and depth integrations the inaccuracies

$$0\left(\frac{\tau}{T}, \frac{T}{\Delta t}\right) \quad and \quad 0\left(\frac{\Delta h}{H}\right)$$

arise, the latter of which can be introduced into the definition of the shear viscosities. The turbulent and shear viscosities are based only on molecular analogy and on requirements for the simplicity of mathematical treatment, without any direct, theoretical support. Nevertheless, in praxis the concept has proved to be quite adequate.

The use of vertical mean velocities implicitly assumes that the velocities at different depths must depend on the same factors and be related to each other. The same applies to the vertical averaging of the turbulent viscosities, too. Thus no clear stratification must be allowed. For the sake of these implicit reasons the weight of this limitation is much more important than the purely mathematical accuracy of it. Under stratified circumstances the equations (15...17) are valid within each particular layer with appropriately modified boundary and shear stresses.

The boundary conditions of equations (15...17) are purely horizontal. Especially at closed boundaries (shore lines) the water flow through the border (i.e. the perpendicular velocity component) must be set equal to zero. At open boundaries the conditions may deal either with water levels (e.g. tides) or with velocities (e.g. river inflows).

21.4 Further Remarks on Different Terms

In order to solve the set of the equations (15...17) a lot of further information is needed of the numerical values of physical coefficients and boundary conditions. In this respect the gravity acceleration g, the coriolis coefficient f and the atmospheric-pressure distribution $p_a(x, y)$ are geographically well known or readily measurable. It is not an essential limitation (and can easily be amended), if we restrict ourselves to such small areas (o(100 km)), where constant values for g, f and p_a can be assumed. In addition to these the bottom configuration of the basin h(x, y) is needed for adequate computation. Similar to the former, the wind shear stress r is mainly an external parameter, not depending on the computed flow.

The bottom shear stress $\vec{\tau}_B$, for its part, depends principally on the bottom roughness, on the type of flow near the bottom which in reality is always turbulent and on the velocity just above the bottom. In 2-dimensional calculations $\vec{\tau}_B$ is assumed to be directed against the mean velocity

$$\vec{r}_B = - r\vec{u} \tag{18}$$

Where the bottom friction coefficient r may depend on the velocity $|\vec{u}|$ on the local depth H, on local roughness and even on assumed velocity distributions. Generally it is not true, not even at completely homogenous conditions that $\vec{v}(x, y, -h)$ would be parallel to $\vec{u}(x, y)$. Thus there are no means to adjust the correct direction to the bottom shear stress, although the correct magnitude can be adjusted. In praxis, however, the results (including the directions) of 2-dimensional calculations are in most cases fairly similar to the field measurements. This can be understood, if due to the vertical and horizontal mixing, i.e. the actual turbulence, the effective bottom friction in reality, too, is greatly directed against the mean velocity.

Under stratified conditions when a separate layer is considered, the relative velocity $u_i - u_{i+1}$ between two neighbouring layers must be introduced, but simultaneously the friction coefficient r is highly reduced, many times indeed $r \to 0$.

The combined eddy viscosity (where the effect of molecular viscosity is negligible) is a dynamical quantity describing the flow and not the fluid. Thus it may considerably vary with space, time, flow, wind velocity, basin dimensions etc. As a first approximation, by lack of observations, it is reasonable to assume υ as constant (in time), homogenous (in space) and horizontally isotropic (in direction).

The horizontal boundary conditions may in principle depend on the computed flow field, or they can be considered together with $g, f, \Delta Pa = \vec{0}, h(x, y)$ and $\vec{\tau}_B$ as a purely external factor.

Thus in the governing equations (15...17) there are 4 quantities left, for which no accurate determination nor theoretical formulae are available, i.e.

- the wind shear stress $\vec{\tau}$

- the bottom shear stress $\vec{\tau}_B$
- the eddy dispersion coefficient $\vec{\nu}$ and
- the boundary conditions at open boundaries

The experimental or approximative means for determining these will be considered in last section of this chapter.

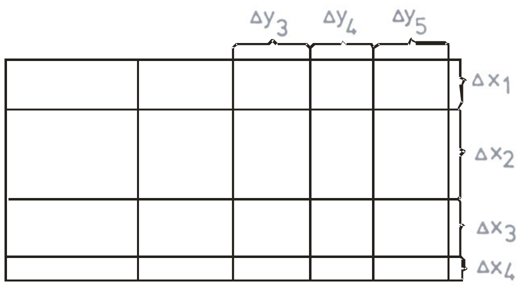
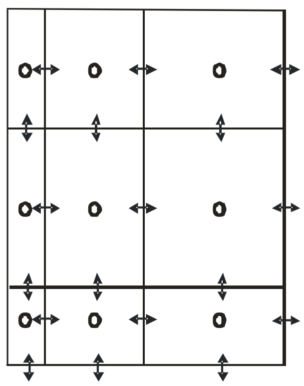


Figure 20. Rectangular grid with variable grid-spacings.



Code:

0 = z-point (central point of control volume for water level)
↔ = u-point (x-boundary of control volume for water level)
↕ = v-point (y-boundary of control volume for water level)

Figure 21. Location of computational points in the space-staggered grid

Figure 22. Placement of computational points in the time-and space-staggered grid.

22 THE NUMERICAL FLOW MODEL

22.1 SOLUTION ALGORITHM FOR INTERIOR POINTS

For description of the bottom topography and for numerical solution of the flow the model area is divided into rectangular elements of irregular size (Figure 20).

When integrating the 2-dimensional basic equations (15... 17 in previous Chapter) over the control volume of each quantity in the space-staggered grid (Figure 21) and denoting

$$U^{k}_{m+\frac{1}{2},n} = \frac{1}{\frac{\Delta x_m + \Delta x_{m+1}}{2} \cdot \Delta y_n \Delta t} \int_{\left(k-\frac{1}{2}\right)\Delta t}^{\left(k+\frac{1}{2}\right)\Delta t} \int_{y_n - \frac{\Delta y_n}{2}}^{y_n + \frac{\Delta y_n}{2}} \int_{x_m}^{x_m + \frac{\Delta x_m + \Delta x_{m+1}}{2}} \cdot udxdydt \tag{19}$$

$$V^{K}_{m,n+\frac{1}{2}} = \frac{1}{\Delta x_m \frac{\Delta y_n + \Delta y_{n+1}}{2} \Delta t} \int_{\left(k-\frac{1}{2}\right)\Delta t}^{\left(k+\frac{1}{2}\right)\Delta t} \int_{y_n}^{y_n + \frac{\Delta y_n + \Delta y_{n+1}}{2}} \int_{x_m - \frac{\Delta x_m}{2}}^{x_m + \frac{\Delta x_m}{2}} \cdot v \, dxdydt \tag{20}$$

$$Z^{k+\frac{1}{2}}_{m,n} = \frac{1}{\Delta x_m \; \Delta y_n \; \Delta t} \int_{k\Delta t}^{(k+1)\Delta t} \int_{y_n - \frac{\Delta y_n}{2}}^{\Delta y_n + \frac{\Delta y_n}{2}} \int_{x_m - \frac{\Delta x_m}{2}}^{x_m + \frac{\Delta x_m}{2}} \eta \, dxdydt \tag{21}$$

$$h_{m,n} = \frac{1}{\Delta x_n \; \Delta y_n} \int_{y_n - \frac{\Delta y_n}{2}}^{y_n + \frac{\Delta y_n}{2}} \int_{x_m - \frac{\Delta x_m}{2}}^{x_m + \frac{\Delta x_m}{2}} hdxdy \tag{22}$$

$$H^{k+\frac{1}{2}}_{m,n} = h_{m,n} + Z^{k+\frac{1}{2}}_{m,n} \tag{23}$$

$$H^{k+\frac{1}{2}}_{m+\frac{1}{2},n} = \frac{\Delta x_m \, H^{k+\frac{1}{2}}_{m+1,n} + \Delta x_{m+1} \, H^{k+\frac{1}{2}}_{m,n}}{\Delta x_m + \Delta x_{m+1}} \tag{24}$$

$$H^{K+\frac{1}{2}}_{m,n+\frac{1}{2}} = \frac{\Delta y_n \, H^{K+\frac{1}{2}}_{m,n+1} + \Delta y_{n+1} H^{K+\frac{1}{2}}_{m,n}}{\Delta y_n + \Delta y_{n+1}} \tag{25}$$

$$U^{K+\frac{1}{2}}_{m,n+\frac{1}{2}} = \frac{\Delta y_n \left(U^{K}_{m-\frac{1}{2},n+1} + U^{K}_{m+\frac{1}{2},n+1} \right) + \Delta y_{n+1} \left(U^{K}_{m-\frac{1}{2},n} + U^{K}_{m+\frac{1}{2},n} \right)}{2\left(\Delta y_n + \Delta y_{n+1}\right)} \tag{26}$$

$$V^{K}_{m+\frac{1}{2},n} = \frac{\Delta x_m \left(v^{K}_{m+1,n-\frac{1}{2}} + v^{K}_{m+1,n+\frac{1}{2}} \right) + \Delta x_{m+1} \left(v^{K}_{m,n-\frac{1}{2}} + v^{K}_{m,n+\frac{1}{2}} \right)}{2\left(\Delta x_m + \Delta x_{m+1} \right)} \tag{27}$$

then the following equations are obtained

$$\begin{aligned}
\frac{U^{K+1}_{m+\frac{1}{2},n} - U^{K}_{m+\frac{1}{2},n}}{\Delta t} &= \frac{-r}{H^{K+\frac{1}{2}}_{m+\frac{1}{2},n}} U^{K}_{m+\frac{1}{2},n} + f\, v^{K}_{m+\frac{1}{2},n} + \frac{\tau_X/\overline{\rho}}{H^{K+\frac{1}{2}}_{m+\frac{1}{2},n}} \\
&- g \frac{Z^{K+\frac{1}{2}}_{m+1,n} - Z^{K+\frac{1}{2}}_{m,n}}{\left(\Delta x_m + \Delta x_{m+1} \right)/2} \\
&+ 2\upsilon \frac{\Delta x_m\, U^{K}_{m+\frac{3}{2},n} - \left(\Delta x_m + \Delta x_{m+1} \right) U^{K}_{m+\frac{1}{2},n} + \Delta x_{m+1}\, U^{K}_{m-\frac{1}{2},n}}{\Delta x_m \left(\Delta x_m + \Delta x_{m+1} \right) \Delta x_{m+1}} \\
&+ 2\upsilon \frac{\frac{\Delta y_{n-1} + \Delta y_n}{2} U^{K}_{m+\frac{1}{2},n+1} - \frac{\Delta y_{n-1} + 2\Delta y_n + \Delta y_{n+1}}{2} U^{K}_{m+\frac{1}{2},n} + \frac{\Delta y_n + \Delta y_{n+1}}{2} U^{K}_{m+\frac{1}{2},n-1}}{\left(\frac{\Delta y_{n-1} + \Delta y_n}{2} \right) \left(\frac{\Delta y_{n-1} + 2\Delta y_n + \Delta y_{n+1}}{2} \right) \left(\frac{\Delta y_n + \Delta y_{n+1}}{2} \right)} \\
&- \delta_{adv} \frac{U^{K}_{m+\frac{3}{2},n} - U^{K}_{m-\frac{1}{2},n}}{\Delta x_m + \Delta x_{m+1}} U^{K}_{m+\frac{1}{2},n} \\
&- \delta_{adv} \frac{U^{K}_{m+\frac{1}{2},n+1} - U^{K}_{m+\frac{1}{2},n-1}}{\frac{\Delta y_{n-1} + 2\Delta y_n + \Delta y_{n+1}}{2}} v^{K}_{m+\frac{1}{2},n}
\end{aligned} \tag{28}$$

$$\begin{aligned}
\frac{V^{K+1}_{m,n-\frac{1}{2}} - V^{K}_{m,n-\frac{1}{2}}}{\Delta t} &= \frac{-r}{H^{K+\frac{1}{2}}_{m,n-\frac{1}{2}}} V^{k}_{m,n-\frac{1}{2}} - f\, U^{K}_{m,n-\frac{1}{2}} + \\
&+ \frac{\tau_Y/\overline{\rho}}{H^{K+\frac{1}{2}}_{m,n-\frac{1}{2}}} - g \frac{Z^{K+\frac{1}{2}}_{m,n} - Z^{K+\frac{1}{2}}_{m,n-1}}{\left(\Delta y_{n-1} + \Delta y_n \right)/2} \\
&+ 2\upsilon \frac{\frac{\Delta x_{m-1} + \Delta x_m}{2} V^{K}_{m+1,n-\frac{1}{2}} - \frac{\Delta x_{m-1} + 2\Delta x_m + \Delta x_{m+1}}{2} V^{K}_{m,n-\frac{1}{2}} + \frac{\Delta x_m + \Delta x_{m+1}}{2} V^{K}_{m-1,n-\frac{1}{2}}}{\left(\frac{\Delta x_{m-1} + \Delta x_m}{2} \right) \left(\frac{\Delta x_{m-1} + 2\Delta x_m + \Delta x_{m+1}}{2} \right) \left(\frac{\Delta x_m + \Delta x_{m+1}}{2} \right)} \\
&+ 2\upsilon \frac{\Delta y_{n-1} V^{K}_{m,n+\frac{1}{2}} - \left(\Delta y_{n-1} + \Delta y_n \right) V^{K}_{m,n-\frac{1}{2}} + \Delta y_n V^{K}_{m,n-\frac{3}{2}}}{\Delta y_{n-1} \left(\Delta y_{n-1} + \Delta y_n \right) \Delta y_n} \\
&- \delta_{adv} \left(\frac{V^{K}_{m+1,n-\frac{1}{2}} - V^{K}_{m-1,n-\frac{1}{2}}}{\frac{\Delta x_{n-1} + 2\Delta x_n + \Delta x_{n+1}}{2}} U^{K}_{m,n-\frac{1}{2}} + \frac{V^{K}_{m,n+\frac{1}{2}} - V^{K}_{m,n-\frac{3}{2}}}{\Delta y_{n-1} + \Delta y_n} V^{K}_{m,n-\frac{1}{2}} \right)
\end{aligned} \tag{29}$$

$$\frac{Z_{m,n}^{K+\frac{3}{2}} - Z_{m,n}^{K+\frac{1}{2}}}{\Delta t} = -\frac{H_{m+\frac{1}{2},n}^{K+\frac{1}{2}} U_{m+\frac{1}{2},n}^{K+1} - H_{m-\frac{1}{2},n}^{K+\frac{1}{2}} U_{m-\frac{1}{2},n}^{K+\frac{1}{2}}}{\Delta x_m} -$$
$$\frac{-H_{m,n+\frac{1}{2}}^{K+\frac{1}{2}} V_{m,n+\frac{1}{2}}^{K+1} - H_{m,n-\frac{1}{2}}^{K+\frac{1}{2}} V_{m,n-\frac{1}{2}}^{K+1}}{\Delta y_n} \qquad (30)$$

where

$U_{m+\frac{1}{2},n}^{K}, V_{m,n-\frac{1}{2}}^{K}, Z_{m,n}^{K+\frac{1}{2}}, H_{m,n}^{K+\frac{1}{2}}$ = the basic finite-difference equivalent to u, v, η and H as defined by (Eqs. 19...23) (m/s, m)

$H_{m+\frac{1}{2},n}^{K+\frac{1}{2}}, H_{m,n-\frac{1}{2}}^{K+\frac{1}{2}}, V_{m+\frac{1}{2},n}^{K}, U_{m,n-\frac{1}{2}}^{K}$ = the corresponding interpolates defined by (Eqs. 24...27) (m, m/s)

$\tau_X/\bar{\rho} = \lambda w w_X$, $\tau_Y/\bar{\rho} = \lambda w w_Y$, the wind shear components (N/m^2)

$r, f, g, v, \lambda, w, w_X, w_Y$ = bottom friction, Coriolis coefficient, gravity acceleration, combined viscosity (including the effects of averaging over time, depth and the control volume), wind friction, wind velocity and its components, respectively.

x_m, y_n = the coordinate of the $Z_{m,n}^{K+\frac{1}{2}}$ point (m)

$\Delta x_m, \Delta y_n$ = the mesh sizes of mth column and nth row respectively (= dimensions of the control volume for $Z_{m,n}^{K+\frac{1}{2}}$) (see Figure 22)

Δt = the time step, (s) and

$$\delta_{adv} = \begin{cases} 0 & \text{if the advective terms are\_neglected} \\ 1 & \text{if they are included} \end{cases}$$

The equations (28…30) can readily be solved for the highest time values

$$U_{m+\frac{1}{2},n}^{K+1}, V_{m,n-\frac{1}{2}}^{K+1}, Z_{m,n}^{K+\frac{3}{2}}$$

yielding an explicit one-step 2-level algorithm /4/.

22.2 Boundary Conditions

Through closed boundaries no water is allowed to be transported, i.e.

$$U_{m+\frac{1}{2},n}^{K+1} = 0 \quad \text{at x-boundaries parallel to the y-axis, } x = x_m + \frac{\Delta x_m}{2} \qquad (31)$$

$$V_{m,n-\frac{1}{2}}^{K+1} = 0 \quad \text{at y-boundaries parallel to the x-axis, } y = y_n - \frac{\Delta y_n}{2} \qquad (32)$$

By drawing the model boundaries through the perpendicular velocity points only, no other conditions are needed. This is a considerable advantage of the grid system used.

Also at open boundaries the Dirichlet-type conditions can be assumed, i.e.

$$H^{K+\frac{1}{2}}_{m+\frac{1}{2},n} \quad U^{K+1}_{m+\frac{1}{2},n} = Q^{X}_{m+\frac{1}{2},n}(t) \quad \text{at x-boundaries (y = constant)} \tag{33}$$

$$H^{K+\frac{1}{2}}_{m,n-\frac{1}{2}} \quad V^{K+1}_{m,n-\frac{1}{2}} = Q^{Y}_{m,n-\frac{1}{2}}(t) \quad \text{at y-boundaries (x = constant)} \tag{34}$$

$$Z^{K+\frac{3}{2}}_{m,n} = S_{m,n}(t) \quad \text{at any boundaries,} \tag{35}$$

where Q^X (m^2/s), Q^Y (m^2/s) and S (m) are externally given values for the x-flow, y-flow and surface level, respectively. As an alternative - if proper estimates for boundary values are not available - they must be extrapolated from the results computed in the interior of the model area (Virtanen 1977 /5/). This, however, results in serious difficulties with the model stability and must be carried out very carefully.

22.3 Formal Accuracy and Consistency

The formal accuracy of each finite-difference term - the order of their truncation error in terms of $\Delta x_m, \Delta y_m, \Delta t$ compared to the corresponding partial derivative, and the consistency in the meaning that the truncation error approaches zero with $\Delta x_m, \Delta x_n, \Delta t \to 0$ - can be checked by Taylor-series expansion /4/.

For the practical point of view the formal accuracy is of little importance compared to the behavioural errors of the finite-difference scheme /4/ but this does not apply to the consistency.

In respect to time the equation of continuity (30) is totally based on central differences, and is thus second order accurate o(Δt^2) - if the use of $H^{K+\frac{1}{2}}$ instead of H^{K+1} is not taken into account (this is only first order accurate o(Δt), but as the water level fluctuations are small compared to the total depth this detail is of no practical meaning). Similarly in the equations of motion (Equations 28, 29) the surface gradient and the wind shear, which are the dominating terms in wind-induced circulation, are strictly of o(Δt^2); whereas the other terms as forward- differences are only first order accurate o(Δt).

Strictly taken the total truncation errors are

$$\begin{aligned}
\varepsilon_z = &- \frac{1}{24}\frac{\partial^3(\mathrm{H}\,U)}{\partial x^3}\Delta x_m^{\;2} - \frac{1}{24}\frac{\partial^3(\mathrm{H}\,V)}{\partial y^3}\Delta y_n^{\;2} - \frac{1}{24}\frac{\partial^3\eta}{\partial t^3}\Delta t^2 \\
&+ \frac{1}{2}\frac{\partial \eta}{\partial t}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)\Delta t \\
&- \frac{1}{2}\frac{\partial^2 \mathrm{H}}{\partial x^2} u\left(\Delta x_{m+1} - \Delta x_{m-1}\right) - \frac{1}{2}\frac{\partial^2 \mathrm{H}}{\partial y^2} v\left(\Delta y_{n+1} - \Delta y_{n-1}\right) \\
&- \frac{1}{4}\frac{\partial^2 \mathrm{H}}{\partial x^2}\left[\frac{\partial u}{\partial x}\Delta x_m\left(\Delta x_{m+1} - \Delta x_{m-1}\right)\right] \\
&- \frac{1}{4}\frac{\partial^2 \mathrm{H}}{\partial y^2}\left[\frac{\partial v}{\partial y}\Delta y_n\left(\Delta y_{n+1} - \Delta y_{n-1}\right)\right]
\end{aligned} \tag{36}$$

$$\begin{aligned}
\varepsilon_u = &\frac{1}{2} + \left[\frac{r}{\mathrm{H}}\frac{\partial u}{\partial t} - f\frac{\partial v}{\partial t} - \upsilon_{XX}\frac{\partial^3 u}{\partial x^2 \partial t} - \upsilon_{XY}\frac{\partial^3 u}{\partial y^2 \partial t} + \right. \\
&\left. + \frac{\partial}{\partial t}\left(u\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial t}\left(v\frac{\partial u}{\partial y}\right)\right]\Delta t \\
&+ \left[\frac{1}{4} g\frac{\partial^2 \eta}{\partial x^2} + \frac{1}{3}\upsilon_{XX}\frac{\partial^3 u}{\partial x^3} - \frac{1}{2}u\frac{\partial^2 u}{\partial x^2}\right]\left(\Delta x_{m+1} - \Delta x_m\right) \\
&+ \left[\frac{1}{6}\upsilon_{XY}\frac{\partial^3 u}{\partial y^3} - \frac{1}{4}v\frac{\partial^2 u}{\partial y^2}\right]\left(\Delta y_{n+1} - \Delta y_{n-1}\right) \\
&+ \frac{1}{8}\left[\frac{r}{\mathrm{H}^2}u\frac{\partial^2 \mathrm{H}}{\partial x^2} + f\frac{\partial^2 v}{\partial x^2} - \frac{\lambda\, WW_X}{\mathrm{H}^2}\frac{\partial^2 \mathrm{H}}{\partial x^2}\right]\Delta x_m\,\Delta x_{m+1} \\
&+ \frac{1}{24}\left[-g\frac{\partial^3 \eta}{\partial x^3} + 2\upsilon_{XX}\frac{\partial^4 u}{\partial x^4} - 4u\frac{\partial^3 u}{\partial x^3}\right]\left(\Delta x_m^{\;2} + \Delta x_m \Delta x_{m+1} + \Delta x_{m+1}^{\;2}\right) \\
&+ \frac{1}{8}\left[f\frac{\partial^2 v}{\partial y^2} - \frac{1}{2}\upsilon_{XY}\frac{\partial^4 u}{\partial y^4} - v\frac{\partial^3 u}{\partial y^3} - \frac{\partial^2 v}{\partial y^2}\frac{\partial u}{\partial y}\right]\Delta y_n^{\;2} \\
&+ \frac{1}{48}\left[-\upsilon_{XY}\frac{\partial^4 u}{\partial y^4} - 2v\frac{\partial^3 u}{\partial y^3}\right]\left(\Delta y_{n-1}^{\;2} + \Delta y_{n-1}\Delta y_{n+1}\right. \\
&\left. + \Delta y_{n+1}^{\;2} + 3\Delta y_{n-1}\,\Delta y_n + 3\Delta y_n\,\Delta y_{n+1}\right) \\
= &\;\sigma\left(\left(\Delta x_{m+1} - \Delta x_m\right),\; \left(\Delta y_{n+1} - \Delta y_{n-1}\right);\; \left(\Delta x_m\,\Delta x_{m+1}\right),\; \left(\Delta x_m^{\;2}\right),\right. \\
&\left(\Delta x_{m+1}^{\;2}\right),\; \left(\Delta y_{n-1}^{\;2}\right),\; \left(\Delta y_{n-1}\,\Delta y_n\right),\; \left(\Delta y_n^{\;2}\right),\; \left(\Delta y_n\,\Delta y_{n+1}\right), \\
&\left.\left(\Delta y_{n+1}^{\;2}\right),\; \left(\Delta y_{n-1}\,\Delta y_{n+1}\right);\; (\Delta t)\right)
\end{aligned} \tag{37}$$

which illustrate the importance of the variable mesh sizes.

As the truncation errors (36), (37) - and ε_v completely analogous to (37) - approach zero with decreasing time- and space-increments $\Delta t, \Delta x_m, \Delta y_n \to 0$, all the finite-difference approximations have been proved to be strictly consistent.

22.4 Stability and Convergence

The stability of a two-dimensional finite-difference scheme- especially with nonlinear terms- can strictly be determined experimentally only. From linearized, simplified and even one-dimensional considerations some useful guidelines may be received.

According to the Lax' equivalence theorem the stability- the non-increase of the round-off error- and the convergence- the approach of the finite-difference solution to that of the partial differential equations- are equivalent in case of *consistent* finite-difference schemes /4/.

The stability of the algorithm for the interior points can be tested e.g. by the v. Neumann stability analysis /4/, where the deviations are expressed as Fourier-series and the stability of each component will be provided. The effects of boundary conditions can be taken into account by the matrix stability analysis. In both cases the norm of the amplification matrix $\vec{\vec{A}}$ is required to be not more than unity

$$\left\| \vec{\vec{A}} \right\| < 1 \qquad \text{("strong" stability)}$$

$$\left\| \vec{\vec{A}} \right\| < 1 + o(\Delta t) \qquad \text{("weak" stability)} \tag{38}$$

As the strict solution of the matrix stability conditions is restricted to extremely small grid nets and to very simple cases, the norm must be estimated upwards or downwards yielding to sufficient conditions (the stability is certain if the conditions are fulfilled) or to necessary conditions (the instability is certain if the conditions are not fulfilled), respectively.

In general the effects of boundary conditions are crucial for the stability- like for the formal accuracy, too- but in case of the Dirichlet conditions, the boundary values do not impair the stability from that of the interior point algorithm. The external wind force does not either have a direct effect on the stability of the present equations.

With equal grid-spacings the v. Neumann stability analysis after neglecting the transversal advection and the coriolis terms- results in several conditions, of which the approximation

$$\Delta t \leqq \frac{1}{\sqrt{g\mathrm{H}_{\max}}} \frac{\Delta_x \Delta_y}{\sqrt{\Delta x^2 + \Delta y^2}} \tag{39}$$

is of most practical interest. The other restrictions are quite similar to those derived in connection with the matrix stability analysis.

The equations (28...30) are so complicated that neither the strict solution nor the upper estimate of the matrix stability conditions render any useful results. The lower estimate, by contrast, gives the necessary but not satisfactory conditions.

$$\Delta t \leqq \left(\frac{r}{\mathrm{H}^{K+\frac{1}{2}}_{m+\frac{1}{2},n}} + \frac{2\upsilon}{\Delta x_m + \Delta x_{m+1}} + \delta_{adv} \frac{U^K_{m+\frac{3}{2},n} - U^K_{m-\frac{1}{2},n}}{\Delta x_m + \Delta x_{m+1}} \right)^{-1} \tag{40}$$

$$\Delta t \leqq \left(\frac{r}{H^{K+\frac{1}{2}}_{m+\frac{1}{2},n}} + \frac{2\upsilon}{\Delta x_m + \Delta x_{m+1}} + \delta_{adv} \frac{U^K_{m+\frac{3}{2},n} - U^K_{m-\frac{1}{2},n}}{\Delta x_m + \Delta x_{m+1}} \right)^{-1} \tag{41}$$

$$\begin{aligned} \delta_{adv}\left(U^K_{m-\frac{1}{2},n} - U^K_{m+\frac{3}{2},n} \right) &\leqq \frac{\Delta x_m + \Delta x_{m+1}}{H^{K+\frac{1}{2}}_{m+\frac{1}{2},n}} r + \\ &+ \frac{\Delta x_m + \Delta x_{m+1}}{\frac{1}{2}\left(\Delta y_{n-1} + \Delta y_n\right) \cdot \frac{1}{2}\left(\Delta y_n + \Delta y_{n+1}\right)} 2\upsilon + \frac{\Delta x_m + \Delta x_{m+1}}{\Delta x_m \, \Delta x_{m+1}} 2\upsilon \end{aligned} \tag{42}$$

$$\begin{aligned} \delta_{adv}\left(V^K_{m,n-\frac{3}{2}} - V^K_{m,n+\frac{1}{2}} \right) &\leqq \frac{\Delta y_{n-1} + \Delta y_n}{H^{K+\frac{1}{2}}_{m,n-\frac{1}{2}}} r + \\ &+ \frac{\Delta y_{n-1} + \Delta y_n}{\frac{1}{2}\left(\Delta x_{m-1} + \Delta x_m\right) \frac{1}{2}\left(\Delta x_m + \Delta x_{m+1}\right)} 2\upsilon + \frac{\Delta y_{n-1} + \Delta y_n}{\Delta y_{n-1} \, \Delta y_n} 2\upsilon \end{aligned} \tag{43}$$

The expressions (40, 41) are usually not restrictive compared to (39). The expressions (42,43), by contrast, are of special importance, because they reveal a source of static instability, which cannot be eliminated by reducing the time step, like in the previous dynamic conditions (39) … (41). This was why the additional advection coefficient δ_{adv} was introduced into the equations (28), (29).

With variable mesh sizes the above expressions, including (39), still preserve their instructive validity, although the strict conditions have not been derived. In order to limit the truncation errors (36, 37) it may be further advisable to restrict the ratio of adjacent increments.

$$\tfrac{1}{2} \leq \frac{x_m}{x_{m+1}} \leq 2 \tag{44}$$

$$\tfrac{1}{2} \leq \frac{y_n}{y_{n+1}} \leq 2 \tag{45}$$

although the instability in the opposite case has not been proved.

22.5 Numerical properties of Finite-Difference Scheme

The inaccuracies of a numerical solution may arise from the truncation of the Taylor-series or from the round-off due to the limited number of bits in the computer. The origin of the errors can be analysed, but it is not as important as the effects of the error on the behaviour of the solution /4/.

The most important dangers in computation of the natural water flow are related to the conservative property that the fluid must neither disappear nor increase in the finite difference scheme and the avoiding of overshoot - that the frictional forces must not turn the flow direction opposite to the original one /4, 5/. The equations (28, 29) are strictly conservative for their most important terms. Only a minor inconservativeness is included in the treatment of the viscous, advective and coriolis terms. The elimination of overshoot is taken into account by the conditions (40, 41).

The amplitude and phase errors are, fortunately, often most important by the less significant short wave lengths only /4, 5/. The transportive errors and the numerical viscosity are mainly due to the advective terms only. In simulation of flow they are usually small compared to the external forces: wind, surface slope and bottom friction. For the sake of the central differences for advective terms, the scheme (28), (29) is not strictly transportive /4/, but in water flow computation the effects of this like several further type of errors can be totally ignored.

23 TRANSPORT AND DISPERSION MODEL

23.1 Fundamental Equations of Transport

The transport and dispersion of any quantity $\tilde{c}$ in a flow field $\vec{\tilde{v}}(x, y, z, t)$ is governed by the transport equation /3/

$$\frac{\partial \tilde{c}}{\partial t} = -\vec{\tilde{v}} \cdot \nabla \tilde{c} + D_m \nabla^2 \tilde{c} \qquad (46)$$

where

∇ = gradient operator (m^{-1})

∇^2 = Laplace operator and (m^{-2})

D_m = (molecular) diffusion coefficient (m^2/s) of the quantity in question.

The flow field $\vec{\tilde{v}}(x, y, z, t)(m/s)$ is the momentaneous local flow velocity of Equation (1) (Section). In order to get the transport Equation (119) consistent with the model velocities of Sections and integrations similar to those of Section and Equations (19)...(21) are subjected to Equation (119). The quantities considered *c* (mg/1) are the concentration of oil in the surface layer and the concentration of dissolved material in the whole water column.

When integrating Equation (119) over the observation timescale T an equation similar to itself is obtained for the smoothed time variation $\bar{c}$ with D_m replaced by the eddy diffusivity tensor $\vec{\vec{D}}_t + D_m.\vec{\vec{I}}$

$$\vec{\vec{D}}_t = \begin{pmatrix} D^t_{xx} & 0 & 0 \\ 0 & D^t_{yy} & 0 \\ 0 & 0 & D^t_{zz} \end{pmatrix} \qquad (47)$$

where

$$D^t_{xx} = -\frac{1}{\mathrm{T}} \int_0^{\mathrm{T}} v'_x \, c' \, dt \Big/ \frac{\partial \bar{c}}{\partial x} \qquad (48)$$

$$D^t_{zz} = -\frac{1}{\mathrm{T}} \int_0^{\mathrm{T}} v'_z c' \, dt \Big/ \frac{\partial \bar{c}}{\partial z} \qquad (49)$$

$$D^t_{yy} = -\frac{1}{\mathrm{T}} \int_0^{\mathrm{T}} v'_y \, c' dt \Big/ \frac{\partial \bar{c}}{\partial y} \qquad (50)$$

$v_{x'}$ $v_{y'}$ v_z = components of $\vec{v}(m/s)$

$\bar{c}$ = average concentration during the observation time-scale (mg/1)

$v'_{x'}$ $v'_{y'}$ $v'_{z'}$ c' = turbulent fluctuations of $\tilde{v}_{x'}$ $\tilde{v}'_{y'}$ $\tilde{v}_{z}, \bar{c}$ respectively around their time-averages $v_{x'} v_{y'}$ $v_{z'} \bar{c}$.

$\vec{\vec{I}}$ = unit tensor $\begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix}$

The integration over the layer depth H further yields

$$\mathrm{H}\frac{\partial c}{\partial t} = -\mathrm{H}\,u\frac{\partial c}{\partial x} - \mathrm{H}\,v\frac{\partial c}{\partial y} + \frac{\partial}{\partial x}\left[\mathrm{H}\left(D_{Hx} + \bar{D}_{tx} + D_m\right)\frac{\partial c}{\partial x}\right]$$

$$+\frac{\partial}{\partial y}\left[\mathrm{H}\left(D_{\mathrm{H}y} + \bar{D}_{ty} + D_m\right)\frac{\partial c}{\partial x}\right] \tag{51}$$

Where *c*, *u* and *v* are now the depth averaged concentration and horizontal velocity components and where the shear fluctuations now result in dispersion coefficient $\vec{\vec{D}}_H$ with its diagonal elements fully analogous with the eddy diffusivities (Equations 123...123), viz.

$$\vec{\vec{D}}_H = \begin{pmatrix} D_{\mathrm{H}x} & 0 \\ 0 & D_{\mathrm{H}y} \end{pmatrix} \tag{52}$$

$$D_{\mathrm{H}x} = -\frac{1}{\mathrm{H}}\int_{-h}^{\eta} u''.c''dz \Big/ \frac{\partial c}{\partial x} \tag{53}$$

$$D_{Hy} = -\frac{1}{\mathrm{H}}\int_{-h}^{\eta} v''.c''dz \Big/ \frac{\partial c}{\partial y} \tag{54}$$

$$a = \frac{1}{\mathrm{H}}\int_{-h}^{\eta} \bar{a}\,dz \tag{55}$$

?? = the depth – averaged value of the eddy filtered quantity a, with $a = u, v$ for $\bar{a} = v_x, \ v_y$ and a = c for $\bar{a} = \bar{c}$

$a'' = \bar{a} - a =$ the shear fluctuation of the quantity a for $a = u, v, c,$ i.e. its deviation from its depth-average, and finally

$$\bar{D}_{tx} = \frac{1}{\mathrm{H}}\int_{-h}^{n} D^t_{xx}\,dz \tag{56}$$

$$\bar{D}_{ty} = \frac{1}{\mathrm{H}}\int_{-h}^{\eta} D^t_{yy}\,dz \tag{57}$$

With these operations the quantities of the transport Equation (124) are consistent with those of the depth-averaged equations of motion (in 16, 15 in the previous chapter).

23.2 Numerical Solution

For numerical solution of the two-dimensional transport Equation (124) the principle of fractional time-steps (the time-splitting method or multistep procedure) /4/ will be utilized. It describes in a realistic way the simultaneous advection and dispersion from the very beginning and can take advantage of the time-scales of both processes independently of each other. With the time-steps Δ_{t_a} for advection and Δ_{td} for dispersion the solution algorithm reads /5/

$$\left.\frac{\partial c}{\partial t}\right|^{k+p.\Delta} = -\frac{1}{\mathrm{H}}\left[\mathrm{H}\,u\frac{\partial}{\partial x} + \mathrm{H}\,v\,\frac{\partial}{\partial y}\right]C\Big|^{k+p.\Delta}$$

for $p = 1,\cdots,\left[\frac{\Delta t_d}{\Delta t_a}/2\right]$ and

$$\text{for } p = \left[\frac{\Delta t_d}{2\cdot\Delta t_a}\right] + 2,\cdots,\left[\frac{\Delta t_d}{\Delta t_a}\right] + 1 \tag{58}$$

$$\left.\frac{\partial c}{\partial t}\right|k + p\cdot\Delta = \frac{1}{\mathrm{H}}\left[\frac{\partial}{\partial x}\left(\mathrm{H}\,D_x\frac{\partial}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mathrm{H}\,D_y\frac{\partial}{\partial y}\right)\right]C\Big|k+p\cdot\Delta$$

$$\text{for } p = \left[\frac{\Delta t_d}{2\cdot\Delta t_a}\right] + 1 \tag{59}$$

Where

[A] = largest integer less than A

$a\big|^{k+p.\Delta}$ = value of the expression a at the time

t = $(k + p.\Delta).\Delta_{ta}$ after the beginning of the simulation

k = integer time-step index

p = fractional time-step index and

Δ = 1 / (1 + $[\Delta t_d / \Delta t_a]$) .

The time-derivatives are estimated with the forward-time differences /4/

$$\left.\frac{\partial c}{\partial t}\right|k + p\cdot\Delta = \frac{1}{\Delta t_p}\left(C^{k+(p+1)\Delta} - C^{k+p.\Delta}\right) + O\left(\Delta t_p\right) \tag{60}$$

where

$$\Delta t_p \begin{cases} = \left(\left[\frac{\Delta t_d}{\Delta t_a}\right] + 1\right)\Delta t_a & whenp = \left[\frac{\Delta t_d}{2\cdot\Delta t_a}\right] + 1 \\ = \Delta t_a & otherwise. \end{cases}$$

For integer values of $k + p.\Delta$ the time-splitting (Equations 131 ... 133) is consistent with the continuous Equation (124) and only these values have a physical meaning.

The advective terms are estimated with the up-wind differences /4/, /5/

$$
\begin{aligned}
&\left(-H u \frac{\partial c}{\partial x} - H v \frac{\partial c}{\partial y}\right)\Bigg|_{m,n}^{k+p\cdot\Delta} = \\
&- \frac{1}{2}\left[H_{m+\frac{1}{2},n}^{k+p.\Delta} \cdot \left(U_{m+\frac{1}{2},n}^{k+p.\Delta} - \left|U_{m+\frac{1}{2},n}^{k+p.\Delta}\right|\right)\right] \cdot \frac{C_{m+1,n}^{k+p.\Delta} - C_{m,n}^{k+p.\Delta}}{\Delta x_m} \\
&+ \frac{1}{2}\left[H_{m-\frac{1}{2},n}^{k+p.\Delta} \cdot \left(U_{m-\frac{1}{2},n}^{k+p.\Delta} + \left|U_{m-\frac{1}{2},n}^{k+p.\Delta}\right|\right)\right] \cdot \frac{C_{m,1,n}^{k+p.\Delta} - C_{m,n}^{k+p.\Delta}}{\Delta x_m} \\
&+ \frac{1}{2}\left[H_{m,n-\frac{1}{2}}^{k+p.\Delta} \cdot \left(V_{m,n-\frac{1}{2}}^{k+p.\Delta} + \left|V_{m,n-\frac{1}{2}}^{k+p.\Delta}\right|\right)\right] \cdot \frac{C_{m,n-1}^{k+p.\Delta} - C_{m,n}^{k+p.\Delta}}{\Delta y_n} \\
&- \frac{1}{2}\left[H_{m,n+\frac{1}{2}}^{k+p.\Delta} \cdot \left(V_{m,n+\frac{1}{2}}^{k+p.\Delta} - \left|V_{m,n+\frac{1}{2}}^{k+p.\Delta}\right|\right)\right] \cdot \frac{C_{m,n+1}^{k+p.\Delta} - C_{m,n}^{k+p.\Delta}}{\Delta y_n} \\
&+ O\left(\Delta X_m, \Delta Y_n\right)
\end{aligned}
\tag{61}
$$

And the dispersive terms with space-centered differences, /4/

$$
\begin{aligned}
&\left[\frac{\partial}{\partial x}\left(HD_x \frac{\partial c}{\partial x}\right) + \frac{\partial}{\partial y}\left(HD_y \frac{\partial c}{\partial y}\right)\right]\Bigg|_{m,n}^{k+p\cdot\Delta} = \\
&\frac{D_x \cdot H_{m,n}^{k+p.\Delta}}{\Delta x_m}\left[d_{m,n}^{k+p.\Delta}(-1,0) \cdot \frac{c_{m-1,n}^{k+p\cdot\Delta} - c_{m,n}^{k+p\cdot\Delta}}{\frac{\Delta x_{m-1} + \Delta x_m}{2}}\right. \\
&\left. + d_{m,n}^{k+p.\Delta}(1,0) \cdot \frac{c_{m+1,n}^{k+p\cdot\Delta} - c_{m,n}^{k+p\cdot\Delta}}{\frac{\Delta x_{m+1} + \Delta x_m}{2}}\right] \\
&+ \frac{D_y \cdot H_{m,n}^{k+p.\Delta}}{\Delta y_n}\left[d_{m,n}^{k+p.\Delta}(0,-1) \cdot \frac{c_{m,n-1}^{k+p\cdot\Delta} - c_{m,n}^{k+p\cdot\Delta}}{\frac{\Delta y_{n-1} + \Delta y_n}{2}}\right. \\
&\left. + d_{m,n}^{k+p.\Delta}(0,1) \cdot \frac{c_{m,n+1}^{k+p\cdot\Delta} - c_{m,n}^{k+p\cdot\Delta}}{\frac{\Delta y_{n+1} + \Delta y_n}{2}}\right] \\
&+ O\left(\Delta x_m - \Delta x_{m-1}, \Delta x_{m+1} - \Delta x_m, \Delta y_{n+1} - \Delta y_n, \Delta y_n - \Delta y_{n-1}, \Delta x_m^2, \Delta y_n^2\right)
\end{aligned}
\tag{62}
$$

where the dispersion cross-section $d_{m,n}^{k+p.\Delta}$ (i, j) has been defined as the common water surface of two grid checks /5/

$$
d_{m,n}^{k+p.\Delta}(i,j) = \min\left\{H_{m,n}^{k+p.\Delta}, H_{m+i,n+j}^{k+p.\Delta}\right\} / H_{m,n}^{k+p.\Delta} \tag{63}
$$

It is to be noted that the total dispersion coefficients D_x, D_y include - in addition to the molecular diffusion D_m, to the eddy diffusion $\overline{D}_t$ and to the vertical shear dispersion D_H - terms $D_{\Delta s}$ from the horizontal shear within the grid check. Their definition is analogous to that of D_H in (Equations 125...128) and then

$$D_x = D_{\Delta s,x} + \overline{D}_{H,x} + \overline{\overline{D}}_{tx} + D_m \tag{64}$$

$$D_y = D_{\Delta s,y} + \overline{D}_{H,y} + \overline{\overline{D}}_{t,y} + D_m \tag{65}$$

where the bar(s) in $\overline{D}_H$ refer to the horizontal grid-check average and in $\overline{\overline{D}}_t$ to the horizontal and vertical grid-check average.

The flow-velocities $U^{k+p.\Delta}_{m+\frac{1}{2},n}$, $V^{k+p.\Delta}_{m,n+\frac{1}{2}}$ are obtained from the flow models. Equations (64, 65) are then solved for $C^{k+(p+1).\Delta}_{m,n}$ at the highest level of time index at each point (m, n), then for the next index $k+(p+2)\Delta$ and so on. The solution procedure is explicit and therefore its stability can be maintained only if /4, 5/

$$\begin{cases} \Delta t_a \leqq 1/\max \left\{\left|A^{k+p\cdot\Delta}_{m,n}\right|\right\} \\ \Delta t_d \leqq 1/\max \left\{\left|B^{k+p\cdot\Delta}_{m,n}\right|\right\} \end{cases} \tag{66}$$

Where $A^{k+p.\Delta}_{m,n}$ is the sum of all the factors multiplying $C^{k+p.\Delta}_{m,n}$ in (Equation 134) and $B^{k+p.\Delta}_{m,n}$ is the same for $C^{k+p.\Delta}_{m,n}$ in (Equation 135). In the simulations e.g. values like Δt_a =120s and Δt_d =1800s have been typically utilized.

The formal accuracy of the solution was indicated in Equations (133…135)

23.3 Initial and Boundary Conditions

To start the simulation with Equations (131), (132) an initial distribution $C^o_{m,n}$ is needed. For point releases this will be.

$$C^o_{m,n} = \begin{cases} \dfrac{M}{\Delta x_m \cdot \Delta y_n \cdot H^0_{m,n}} & at\_the\_loading\_place \\ 0 & elsewhere \end{cases} \tag{67}$$

when

M = amount released (kg)

$H^o_{m,n}$ = depth of the layer receiving the discharge (m)

Thus the finite-difference model assumes an immediate mixing into the whole grid-check volume. During the time when the horizontal extension of the discharge (e.g. a dye patch) is less than the grid-spacing Δx_m, Δy_n its advection can be traced in the flow field as a solid particle. Its dispersion must then be estimated with analytical expressions.

On the closed boundaries of the model area no concentration is allowed to be exchanged with the shore. The advective exchange (Equation 134) is blocked by the boundary condition of the flow models (Equations 31, 32), the dispersive exchange (Equation135) is blocked by the dispersion cross-sections (Equation136) /5/.

On the open boundaries the extreme possibilities in the dilution are a) immediate dilution into the immense water masses of the Red Sea, i.e.

$$C_B^{k+p\Delta} = O \tag{68}$$

and b) dilution with the same rate as within the model area which results in exponential decrease near the boundary

$$C_B^{k+p\cdot\Delta} = C_{I1}^{k+p\cdot\Delta} \cdot \min\left\{ \left(C_{I1}^{k+p\cdot\Delta} / C_{I2}^{k+p\cdot\Delta}\right)^{\Delta s_1/\Delta s_2},\ 1\right\} \tag{69}$$

where the subscripts

B	= finite-difference space coordinates (m, n) of the open-boundary point
I1, I2	= coordinates of the grid-point locating one and two grid-spacings, respectively, from point B towards the interior of the model area perpendicularly to the boundary
$\Delta s_1, \Delta s_2$	= grid-spacing between the points B...I1 and I1…I2, respectively.

The expected dilution lies somewhere between the two extremes and in the model the following condition was used:

$$C_B^{k+p\cdot\Delta} = \lambda_r \cdot C_{B_2}^{k+p\cdot\Delta} + \left(1-\lambda_r\right) C_{B_1}^{k+p\cdot\Delta} \tag{70}$$

where C_{B2} is obtained by (Equation142), C_{B1} by (Equation 141) and λ_r = (0...) 0,5...0,8. The solution is relatively insensitive to λ_r which indicates that the model area has been sufficiently wide for the transport simulation.

24 DETERMINATION OF PHYSICAL PARAMETERS

24.1 WIND SHEAR STRESS

The wind shear stress $\vec{\tau}_s$ can be directly measured or calculated on the basis of wind velocity profiles, Bengtsson 1973 /9/. Most commonly the stress is related to wind velocities at fixed height above the water surface by Lick 1976, /10/.

$$\vec{\tau}_s / \rho = \lambda \left|\vec{W}\right|^{n-1} \vec{W} \tag{71}$$

where

λ	= wind friction coefficient = $\rho_a C_D / \rho$
C_D	= wind drag coefficient
ρ_a	= density of air (kg/m^3)
$\vec{W}$	= wind velocity vector at given height (m/s)
$n \approx 2.$	

The values of the parameters λ and n will depend on the measuring height and on the roughness of the water surface, which further is a function of wind. Due to these the scattering of the reported values for λ (with n = 2 fixed) has ranged from 1.6•10^{-16} (Elomaa /11/, Virtanen et al. /12/), 1.875 10^{-6} (Bennett /13/), 2•10^{-6} (Sarkkula and Virtanen /14/) and 2.06•10^{-6} (Haq et al. /15/) to 3.2-10^{-6} (Sundermann /16/) and 3.75•10^{-6} (Simons /17/).

In addition to the parameter values the wind velocity itself is a further source of uncertainty, because direct measurements above the water surface are seldomly available. The meteorological observations often take place on-shore and the local variation of wind above the sea surface is usually unknown.

Here the wind data observed in the harbour was directly used as the forcing function. For wind friction its lowest estimate λ = 1.6•10^{-6} was used in order not to overestimate the water exchange.

24.2 BOTTOM SHEAR STRESS

The form and direction of the frictional bottom shear stress $\vec{\tau}_\beta$ has already been fixed by the assumption (18). Thus further consideration shall concern the determination of the bottom friction r only.

For the friction coefficient numerous theoretical formulae have been presented, Starosolszky /18/, of which that of manning has been the most popular one

$$r(\mathrm{H}, u) = \left(n^2 \cdot g / \sqrt[3]{R}\right) \cdot \vec{v}\left(x, y, -h\right) \tag{72}$$

where

n = Kutter roughness coefficient $\left(s.m^{-1/3}\right)$

R	= hydraulic radius of the cross-section (m) (= wet area divided by the wet circle, i.e. $R \approx H$
g	= gravity acceleration $\left(m/s^2\right)$
$\vec{v}(x, y, -h)$	= the bottom velocity (m/s)

The Kutter roughness coefficient *n* is related to the Nikuradse absolute roughness $\Delta \quad (m) \quad by \quad /18/$

$$n_0 / n = 1g(\Delta_0/\Delta) \tag{73}$$

where $n_0 = 0.05\ s.m^{-1/3}$, $\Delta_0 = 915\ cm$, and $\Delta(m)$ is equivalent to the diameter of the grains of the bottom irregularities. The n-values for different bed materials have been tabulated, too /18/. These theoretical formulae are mainly used in river conditions only.

In two-dimensional flow a more direct correlation to H and u is usually looked for. The experimental methods to determine this correlation may be based e.g. on measuring the flow velocities at several stationary wind conditions, Noye 1977, /19/; on measuring the steady-state surface slope under known surface stress; or on the dumping of the amplitude of the surface waves

Of the practical forms of the bottom friction, the depth- independent quadratic friction law

$$r(H, u) = r_1\ u \tag{74}$$

is most similar to Manning's formula (Equation. 145) and superior in its frequency /10/. Earlier the proportionality to the water transport Hu was used, too

$$r(H, u) = r_2 \cdot H\ u \tag{75}$$

even with r_2 = f /5 /16/; and recently the linear depth-independent friction law

$$r(H, u) = r_3 \quad (= \text{constant}) \tag{76}$$

has proved its applicability as well, especially in shallow water systems /19/, /14/. In areas with considerable depth variations a friction coefficient inversely proportional to depth has been introduced

$$r(H, u) = r_4 H^{-1} \tag{77}$$

/15/, /5/ which overestimates the $H^{-1/3}$-dependence of Manning's formula. In general the numerical values of $r(H, u)$ reported in literature are of the order of magnitude of 1 mm/s, although adequate comparison is considerably hampered by the scattering of the functional forms, bottom types, depths and flow velocities. In Finnish and Australian lakes, it is consistently observed for

$$u \le 10\ cm/s$$

$$H \sim 3 \cdots 30\ m$$

$$r_4 = H\ , r(H, u) = 1.5 \quad m\ .\ mm/s$$

/5/,/19/, while in the North-American Great Lakes the value of $r_4 = 5\ m\ mm/s$ vas used by /15/. If the bottom roughness varies considerably at different areas, then all the coefficients $r_1, \cdots r_4$ can further depend on the location similarly to the variation of n (Equation 146) in Manning's formula. In praxis, when the depths and follow velocities are of the same order of magnitude in most parts of the recipient, the difference between Equations (147...150) is not very crucial and the experimental test between them may not be sensible enough. Here the linear friction rule (Equation 149) will be accepted with $r_3 = 0{,}2\ mm/s$.

24.3 Dispersion Coefficients

The determination of the turbulent viscosities can principally be carried out by measuring the actual time series of the velocity components at fixed points, and thereafter calculating the viscosity directly on the basis of the governing equations (5):

$$\nu_{xx}^{t} = -\left[\frac{1}{T}\int_{o}^{T}\left(\tilde{\nu}_x - \nu_x\right)\left(\tilde{\nu}_x - \nu_x\right)dt\right] / \frac{\partial \nu_x}{\partial x} \qquad \text{etc.} \tag{78}$$

Instead of the fixed point velocity measurements, the position of floating buoys can be determined /20/. The relative distance between the buoys then gives an estimate of the length scale of the turbulent eddies influencing the dispersion. Using the observation of the vertical velocity profiles, the total dispersion coefficient can be determined similarly to (Equation 151). Poor accuracy, danger of zero denominator and the tedious field work required are the main drawbacks of this type of definitions.

By assuming the dispersion of velocities equivalent to the dispersion of passive concentrations, i.e. the combined eddy viscosity $\vec{\vec{\nu}}$ equivalent to the dispersion coefficient $\vec{\vec{D}}$ the tracer experiments can be used for determining the desired parameters. In steady flow a continuous tracer release is available. For calculating the transversal exchange coefficients several formulae have been proposed, e.g. Holley et al. 1972, /21/.

$$D_y = \delta' \cdot \frac{\partial}{\partial x}\int_{o}^{B}\int_{-h}^{\eta} \nu_x \cdot \bar{c} \cdot \left(y - y_1\right)^2 \quad dzdy \tag{79}$$

Where

δ'	= coefficient $\approx (1/2\int_0^B\int_{-h}^{\eta} cdzdy)^{-1}\ (kg^{-1})$
B	= the breadth of the reservoir (m)
Y_1	= the position (m) of the maximum vertically integrated concentration

$$c\left(x, y_1\right) = \int_{-h}^{\eta} \bar{c}\left(x, y, z\right) \quad dz \leq c\left(x, y_1\right);\ or/22/:$$

$$D_Y = -\frac{\int_{Y_e}^{Y_r} uc \cdot \left(y - y_1\right) \quad \tan\alpha \cdot dy}{\int_{Y_e}^{Y_r} cdy + c\left(y_r\right)\cdot\left(y_r - y_1\right) + c\left(y_e\right)\cdot\left(y_1 - y_e\right)} \tag{80}$$

where

α	= the angle (rad) between mass flux constant line and the streamline, and

$y_{e'}$ y_r = the practical extreme coordinates (m) of the tracer plume, usually taken as $c(y_e) \approx c(y_r) \approx (0.03 \cdots 0.05).c(y_1)$.

The longitudinal two-dimensional dispersion coefficients D_x are not easy to determine on the basis of continuous tracer release. Usually this type of measurements are applied in river conditions only, where the effect of D_x can be neglected.

In the case of instantaneous tracer release, the dispersion coefficients are determined on the basis of the variances of the tracer patch /23/

$$S_x^2 = \frac{1}{M} \int \int_{-\infty}^{\infty} \int (x - x_o)^2 \, \bar{c} \, dxdydz \tag{81}$$

$$S_y^2 = \frac{1}{M} \int \int_{-\infty}^{\infty} \int (y - y_o)^2 \, \bar{c} \, dxdydz \tag{82}$$

Approximately by

$$D_x = S_x^2 / 2T \tag{83}$$

$$D_y = S_y^2 / 2T \tag{84}$$

where

M = the amount of tracer released = $\int \int_{-\infty}^{\infty} \int \; cdxdydz$ (kg)

T = the time elapsed since the discharge (s), and $x_0, y_0 =$ the center of gravity of the tracer patch (m)

$$\int \int_{-\infty}^{\infty} \int / \, x - x_0 \, / \; cdxdydz = \int \int_{-\infty}^{\infty} \int / \, y - y_0 \, / \, c \;\; dxdydz = 0.$$

Principally the time-dependence and numerical coefficients of (Equations 156, 157) are based on the assumption of Gaussian type distribution of concentration, but as a rule for calculation they can be generally used.

Finally, if no measurements are available, some appreciative formulae have been developed, e.g. /24/:

$$\vec{\vec{D}} = \vec{\vec{\delta}} \, . \, R \, . \, U^* \quad ; \tag{85}$$

Where

U* = the shear velocity ($\approx \sqrt{gRs} \; in \; rivers$) (m/s) s

S = the surface slope $= \Delta \eta \, / \Delta x = \partial \eta \, / \partial x$

R = the hydraulic radius (m), and

$\vec{\vec{\delta}}$ = a dimensionless coefficient matrix.

For two-dimensional dispersion mainly in river conditions $\delta_x \approx 6 - 50 \; and \;\; \delta_y \approx 0.2 - 1.5$. In lakes and reservoirs the application of (Equation 158) is not so straightforward and the difference between the two components not so striking; often a complete isotropy has been assumed.

The values of the dispersion coefficients $\vec{\vec{\upsilon}}$ or $\vec{\vec{D}}$ depend on the length scale L of the eddies /23/. In the case of dispersing tracer patch the dependence

$$D = A \cdot L^m \tag{86}$$

with $A = constant = D_0 / L_0^m \quad and \quad m = 4/3$

can be theoretically derived /23/, /3/. In praxis, slightly lower values m = 1 ... 1.33 have been observed /25/, /26/.

For the numerical values of horizontal dispersion coefficients, quite similar results have been reported from all over the world, i.e. corresponding to

$$L \approx 1\,km$$

$$D_x \approx D_y \approx 1\,m^2/s.$$

These include the North American Great Lakes /23/, Murthy and Okubo /25/, some coastal bays of North America, Laevastu /27/, Leendertse and Liu /28/, the North Sea, Ronday /29/, /3/, the Baltic Sea, Kullenberg /29/, and its Bays /26/.

In transport simulations the dispersion coefficient corresponding to the assumed length scale of the geometry and its eddies was used, viz. $D_x = D_y = 0.04 \cdots 0.4\,m^2/s \quad for\ L = 0.008 \cdots 0.5\,km$. In the flow model, however, the stability of the boundary conditions and a reasonable convergence of the solution required slightly overestimated values for the eddy viscosity, viz. $\upsilon = (1\cdots)10(\cdots100)m^2/s.$ All these values are well below the limit $\upsilon \Delta t / \Delta s^2 < 0.02$ which has been found acceptable for reliable flow determination /5/.

24.4 Open Boundaries

The boundary values for open boundaries can be obtained from

- Field measurements, Abbott et al. /31/
- Physical models, Svensson /32/
- Large scale numerical models /10/, Adam /33/
- Theoretical assumptions, like analytical type conditions at infinity,
- Waldrop and Farmer /34/
- The physical theory of tides /29/, Heaps /35/, /27/, /28/

Sometimes, if none of these means are available, the boundary values must be determined by extrapolation on the basis of the values calculated at internal points /5/.

The currents induced by the sea level rise (and decrease, respectively), most of often determined by tides, are often used in coastal applications.

24.5 Time Behaviour of Flow

The model simulation for each flow field was started with a so-called "cold start", i.e. from still and level water body at the beginning of computation. Thereafter the flow started to develop with time forced by the external conditions. Under steady conditions a steady-state flow - independent of the initial situation - finally resulted in. With the most common eddy viscosity $\upsilon = 10\,m^2/s$ the steady-state (fluctuations less than 2 per cent) was reached in several hours. The approach towards the steady-state was roughly exponential. Thus the largest fluctuations took place at the beginning of computation, and after two hours' simulation the deviations from steady-state values were usually less than $\pm\ 15$ per cent. This period is short compared with the more or less regular diurnal variation observed in the external conditions (see Chapter 22). Thus the model transport with the stepwise changed steady-state flow was almost identical with that produced by the true time-dependent flow.

Eddy viscosities less than $1\ m^2 / s$ will enlarge the period of transient fluctuations but do not seriously affect the 2steady-state flow field for $\upsilon \leq 0.02 \frac{\Delta s^2}{\Delta t} \approx 50 m^2 / s\ /5/$ .In the present case very low eddy viscosities were not possible because of the wide open boundaries. Elsewhere it has been shown that even for $\upsilon = 0$ the average transport during transient period is very near to that driven by steady-state flow /12/, / 36/.Therefore the former procedure was mainly used in transport simulations under fluctuating conditions

25 REFERENCES OF THE MODEL DESCRIPTION CHAPTERS

/1/ Imatran Voima Ltd (1982). Physical Sale Model Study of the Application Area, Sept. 1982.

/2/ Daily, James W ., and Harleman, Donald R.F., (1966), Fluid Dynamics. Addison-Wesley Publishing Company Inc .,Reading, Mass., U.S.A., 454 p.

/3/ Nihoul, Jacques, C.J., (ed.), (1975), Modeling of Marine Systems, Elsevier Oceanography Series No. 10, Elsevier Scientific Publishing Company, Amsterdam Oxford-New York 1975, 272 p.

/4/ Roache, Patrick J. (1974), Computational Fluid Dynamics, Hermosa Publishers, Albuquerque 1974, 375 p.

/5/ Virtanen, Markku (1977), Vesiston virtausten ja sekoittumisen kuvaaminen kaksidimensioisella matemaattisella mallilla (Describing the water flow and mixing by means of a two-dimensional mathematical model, in Finnish), Thesis presented for the degree of Lisenciate in Technology, Technical University of Helsinki, Department of Technical Physics, Espoo 1977, 128 p.

/6/ Ambjorn, Cecilia, Luide, Tiina, Omstedt, Anders and Svensson, Jonny (1981), En operationell oljedriftsmodell for Norra bstersjon (An operational oil drift model for the Northern Baltic, in Swedish), Swedish Meteorological and Hydrological Institute, SMHI Report RH029, Hydrology and Oceanography series, SMHI, Norrkoping 1981, 47 p.

/7/ Svensson, Urban (1979), The Structure of the Turbulent Ekman-layer, Tellus, 31, pp. 340-350

/8/ Bye, J. (1965), Winddriven Circulation in Unstratified Lakes, Limnol. Oceanography, 10, pp. 451-458.

/9/ Bengtsson, Lars (1973), Model studier av circulationsprocesser i sjoar (Model studies of the circulation process in lakes, in Swedish), Lunds Universitetet, Institutionen for vattenbyggnad, Bulletin series A, No. 22, Lund 1973, p. 157.

/10/ Lick, Wilbert, J. (1976), Numerical Models of Lake Currents, U.S. Environmental Protection Agency, Ecological Research Series, EPA-600/3-76-020, Duluth, Minnesota, April 1976, p. 140.

/11/ Elomaa, Esko (1979), Drag coefficient over a lake surface in near-neutrally stratified conditions determined from wind profiles, Nordic Workshop on Lake Dynamics, Lillehammer, Norway, March 28-30, Norwegian Committee for Hydrology, Report No. 2, pp. 333-341.

/12/ Virtanen, Markku, Forsius, John and Sarkkula, Juha (1979), Measured and Modeled Currents of a Lake, Aqua Fennica, 9, p. 3-15.

/13/ Bennett, John R. (1977), A Three-dimensional Model of Lake Ontario's Summer Circulation, I. Comparison with Observations, Journal of Physical Oceanography, Vol. 7, No. 4, July 1977, pp. 591-601.

/14/ Sarkkula, Juha, and Virtanen, Markku (1978), Modeling of water exchange in an estuary, Nordic Hydrology, Vol. 7, No. 1, 1978, p. 17.

/15/ Haq, Aminul; Lick, Wilbert J. and Sheng, Y. Peter (1975), On the time-dependent flow in a lake, Journal of Geophysical Research, Vol. 80, No. 3, January 1975, pp. 431-437.

/16/ Sündermann, Jürgen (1966), Ein Vergleich zwischen der analytischen and der numerischen Berechnung winderzeugter Stromungen and Wasserstande in einem Moeller mitt Anwendungen auf die Nordsee, Mittelungen des Instituts fur Meereskunde der Universitat Hamburg, Nr. IV, Hamburg 1966, p. 77.

/17/ Simons, T.J. (1975), Verification of numerical models of Lake Ontario, part 2, stratified circulations and temperature changes, Journal of Physical Oceanography, Vol. 5, No. 1, January 1975, pp. 98-110.

/18/ Starosolszky, Ödon (1975), The fundamentals of engineering hydraulics, International post-graduate course on Hydrological Methods for Developing Water Resources Management, VIZDOK, Budapest, 75/42, p. 154.

/19/ Noye, B.J. (1977), Wind-induced circulation and water level changes in lakes, Preprints of the International Conference on Applied Numerical Modeling, University of Southampton 11-15 July 1977, Pentech Press Limited, London-Plymouth, 1977, pp. 135-146.

/20/ Eriksson, Erik, and Sundberg, Karin (1973), Turbulence Characteristics of Horizontal Currents in a Small Lake and Their Applications, Hydrology of Lakes Symposium, July 1973, IAHS-AISH Publication No. 109, pp. 335-345.

/21/ Holley, E.R., Simons, J. and Abraham, G. (1972), Some Aspects of Analyzing Transverse Diffusion in Rivers, Journal of Hydraulic Research, Vol. 10, pp. 27-57.

22/ Somlyódi, László (1977), Dispersion measurements on the Danube ,Water Research, Vol. 11, pp. 411-417.

/23/ Murthy, C.R. (1973), Horizontal diffusion in lake currents, Hydrology of Lakes-Symposium, Helsinki, July 1973, IAHS-AISH Publication *No.* 109, pp. 327-334.

/24/ Elder, J.W. (1959), The dispersion of marked fluid in turbulent shear flow, Journal of Fluid Mechanics, Vol. 5, pp. 544-560.

/25/ Murthy, C.R., and Okubo, A. (1977), Interpretation of diffusion characteristics of oceans and lakes appropriate for numerical modeling, Symposium of Modeling of Transport Mechanisms in Oceans and Lakes, Manuscript Report Series *No.* 43, Marine *Environmental* Data Service, Ocean and Aquatic Sciences, Department of the Environment, Ottawa,Ontario, 1977, pp. 129-135.

/26/ Kämäräinen, Veikko J., and Virtanen, Markku (1976), Ydinvoimalaitosten aiheuttama vesiston lampeneminen, Technical Research Center of Finland, Reactor Laboratory, Report *No.* 34, Espoo, Finland, March 1976, 43 p.

/27/ Laevastu, Taivo (1974), A vertically integrated hydro dynamical numerical model (W. Hansen type); model description and operating/running instructions. Part 1., *Environmental* Prediction Research Facility, Monterey, California, AD-778 619. January 1974, p. 63.

/28/ Leendertse, Jan J., Liu, Shiao-Kung (1975), A three-dimensional model for estuaries and coastal seas: vol. II, aspects of computation, Rand Corporation, Santa Monica, R-1762-OWRT, June 1975, p. 123.

/29/ Ronday, F.C. (1974), Mesoscale effects of the "tidal stress" on the residual circulation of the North Sea, ICES Special Meeting on Models of Water Circulation in the Baltic, Pap. No. 16, Copenhagen, September 1974, p. 24.

/30/ Kullenberg, Gunnar (1972), Apparent horizontal diffusion in stratified vertical shear flow, Tellus, Vol. 24, No. 1, 1972, pp. 17-28.

/31/ Abbott, M.B., Bertelsen, J.A., and Warren, I.R. (1974), Applications to the mathematical modeling of the Danish Belts, ICES Special Meeting on Models of Water Circulation in the Baltic, Pap. No. 7, Copenhagen, September 1974.

/32/ Svensson, Urban (1976), HH-leden - strbmningsundersokning i bresund med matematisk model (HH-passage, an investigation of currents in bresund by means of mathematical models, in Swedish), Lund Institute of Technology, Department of Water Resources Engineering, Rapport, No. 3001, Lund, 1976, p. 32.

/33/ Adam, Yves (1977), Modeling the evolution of patches of pollutants in the southern North Sea, Applied Mathematical Modeling, Vol. 1, No. 4, March 1977, pp. 170-176.

/34/ Waldrop, W., and Farmer, R. (1970), Three-dimensional computation of boyant plumes, Journal of Geophysical Research, Vol. 79, No. 9, March 1970, pp. 1269-1276.

/35/ Heaps, N.S. (1972), on the numerical solution of the three-dimensional hydro-dynamical equations for tides and storm surges, Memoirs Societe Royale des Sciences de Liege, 6e series, tome II, 1972, pp. 143-180.

/36/ Sarkkula, Juha, and Virtanen, Markku (1982), Application of mathematical flow field in Finland, Swedish IHP -Meeting on the Application of Flow Models in Lakes and Coastal Areas, Stockholm, 15-16 November, Swedish Natural Science Research Council, Hydrology Committee Report. P.19.

26 TURBULENCE MODELS

(*The text in this chapter is obtained from a GOTM (General Ocean Turbulence Model) publication.*)

A standard approach for turbulence parameterization is the so-called eddy viscosity principle, which has been adopted here. It relates the turbulent fluxes to the gradient of the transported property by means of an eddy viscosity (for momentum) or eddy diffusivity (for tracers).

For calculating eddy viscosity/diffusivity, we apply for all turbulence models presented here the relation of Kolmogorov [1942] and Prandtl [1945], which relates these turbulent exchange coefficients to the product of a velocity scale and a length scale:

$$\upsilon_t = c_\mu \sqrt{k}L \tag{87}$$

and

$$\upsilon_t' = c_\mu' \sqrt{k}L, \tag{88}$$

where k is the turbulent kinetic energy (TKE) in $Jkg^{-1} = m^2s^{-2}, L$ the turbulent macro length scale in m and c_μ and c_μ' are the dimensionless so-called stability functions.

For the relations (4) and (5), different forms are found in the literature. In the well known publications of Mellor and Yamada [1974, 1982], the TKE is substituted by the turbulence intensity $q = \sqrt{2k}$. Therefore, the eddy viscosities and stability functions in (4) and (5) correspond to $K_M, K_H, S_M, and\ S_H$ used in Mellor and Yamada [1974, 1982] in the following way:

$$K_M = \upsilon_t \quad K_H = \upsilon_t' \quad S_M = c_\mu / 2^{1/2} \quad S_H = c_\mu' / 2^{1/2}. \tag{89}$$

In the $k - \varepsilon$ literature (see Rodi [1980, 1987]), the length scale is substituted by the normalized dissipation rate which has here the unit $Wkg^{-1} = m^2s^{-3}$:

$$\varepsilon = \left(c_\mu^0\right)^3 \frac{k^{3/2}}{L} \tag{90}$$

with the constant $c_\mu^0 = 0.5562$. This has the consequence that the stability functions in the literature of Rodi [1980, 1987] are by the factor of $\left(c_\mu^0\right)^3$ smaller than those given here in this report. The dissipation rate ε is also an interesting physical quantity in the field of turbulence, since it can be directly measured (see e.g. Simpson et al. [1996], Prandke and Stips [1996]).

In principle, k, L and the stability functions can be calculated independently from each other. Therefore, we designed a section for each of them. It should however be considered that standard turbulence models including each a set of $k, L, c_\mu\ and\ c_\mu'$

are calibrated and even only slight deviations from that might have the consequence that the calibration is not valid any more. For the TKE calculation see section 22.1, for the length scale calculation see section 22.2, and for the stability functions see section 22.3.

Other important parameters in turbulence modelling are the shear production

$$P = \upsilon_t M^2 \tag{91}$$

and the buoyancy production

$$B = -\upsilon_t' N^2 \tag{92}$$

with the shear frequency M from (2.21) and the BruntVäisälä frequency N defined by

$$N^2 = \partial_z b. \tag{93}$$

26.1 Turbulent kinetic energy

Most of the models found in literature are based on the same transport equation for turbulent kinetic energy. They differ mainly in the treatment of the eddy diffusivity for the TKE. The full transport equation is introduced in section 25.1.1, its algebraic idealization in section 25.1.2.

26.1.1 Transport equation

A three-dimensional form of the TKE equation will not be given here, because even in many three-dimensional models, turbulence models are used as local water column models where horizontal advection and diffusion as well as vertical advection are neglected. Such terms would only be of importance near sharp fronts which cannot be resolved in most of the present models. The reason for this is that the time scales of turbulence are generally much shorter than those of the mean flow which drives advection and horizontal diffusion. In the one-dimensional form, the TKE equation reads then as:

$$\partial_t k - \partial_z \left(\upsilon_k \partial_z k\right) = P + B - \varepsilon, \tag{94}$$

Under only a few modelling assumptions, this equation can be derived from the NavierStokes equations (for a full derivation see e.g. Rodi [1980]). The eddy diffusivity relevant for the TKE is calculated as $\upsilon_k = \upsilon_t$ for the $k - \varepsilon$ and as $\upsilon_k = 0.283\sqrt{kL}$ for the MellorYamada model.

Two theoretically equivalent sets of boundary conditions for k, Dirichlet and flux conditions can be derived from the logarithmic law of the wall. The Dirichlet conditions are easily derived from the assumption $P = \varepsilon$ in the boundary layers. By using the definition of the bottom friction velocity u_*^b and combining it with the relation of Prandtl and Kolmogorov (4) and (5) and the definition of (2.34), this boundary condition can be written as:

$$k = \left(\frac{u_*^b}{c_\mu^0}\right)^2, \qquad z = -H. \tag{95}$$

The analogue surface boundary condition reads as

$$z = -H. \tag{96}$$

$$k = \left(\frac{u_*^s}{c_\mu^0}\right)^2, \qquad z = \zeta \tag{97}$$

with the surface friction velocity $u_*^s = \left(\left(\tau_x^s/\rho_0\right)^2 + \left(\tau_y^s/\rho_0\right)^2\right)^{1/4}$.

Assuming that there is a boundary layer each at the surface and at the bottom with $P = \varepsilon$ and a constant in space value of k, leads to the noflux conditions

$$\upsilon_k \partial_z k = 0, \qquad z = -H \tag{98}$$

and

$$\upsilon_k \partial_z k = 0, \qquad z = -H \tag{99}$$

Generally, the Dirichlet conditions are used with the MellorYamada, and the flux conditions with the $k-\varepsilon$ model. An advantage of the flux boundary condition for k is the easy treatment of turbulence near the stressfree surface. The Dirichlet conditions would lead to an eddy viscosity profile which is not decreasing towards the surface. The choice of the boundary conditions is even more important for the length scale equation. The current version of GOTM allows for both sets of boundary conditions. It should be noted here that the effect of breaking surface waves on near surface mixing is not yet included in GOTM. Following the approach from Craig and Banner [1994], a flux of turbulent kinetic energy proportional to the surface friction velocity cubed could be considered for this purpose. They obtained a good approximation to measured near surface dissipation rates with a oneequation MellorYamada model, where the near surface macro length scale increases linearly with depth. Our experiments with the twoequation MellorYamada model and with the k-model gave such different results, that we were reluctant to include the Craig and Banner [1994] approach at this stage of GOTM.

26.1.2 Algebraic form

From the local equilibrium assumption,

$$P + B = \varepsilon, \tag{100}$$

i.e., that production and dissipation of turbulent kinetic energy are in balance, an lgebraic form for calculating TKE can easily obtained by using (2.34), (2.31), and (2.32). Together with a lower limit which is necessary in order to avoid unphysical negative values for the turbulent kinetic energy, k is of the following form:

$$k = \max\left\{\frac{L^2}{\left(c_\mu^0\right)^3}\left(c_\mu M^2 - c_\mu' N^2\right), k_{\min}\right\}. \tag{101}$$

26.2 Turbulent length scale

The turbulent length scale has always been subject to empirical considerations. One of its properties which is agreed about among turbulence modellers, is the linear decrease towards a fixed boundary:

$$L = k\left(\tilde{z} + z_0\right) \tag{102}$$

with the von Kármán constant $k = 0.4$, the distance from the boundary, $\tilde{z}$, and the roughness length of the boundary, z_0 . All length scale parameterizations described here will follow that law near fixed boundaries, most of them as well near the surface. It should be already noted here, that the $k - \varepsilon$ model is the only one which automatically (by only fixing boundary conditions) fulfils this law.

26.2.1 Transport equations

In oceanography, mainly two different transport equations for a length scale related variable are in use, the ε equation in the $k - \varepsilon$ and the kL equation in the Mellor-Yamada model. The structural similarity of these two equations has been shown by Petersen [1996], Burchard et al. [1998] and Burchard and Petersen [1999].

Since the late seventies, a controverse discussion about the higher physical relevance of each of the two length scale equation has been carried out. The arguments of the main protagonists of this discussion are cited here:

Mellor and Yamada [1982]:

> *While one cannot assert great confidence in [the kL equation], we prefer it rather than the differential equation for dissipation ...*
>
> *... it seems fundamentally wrong to us to use an equation which describes the small scale turbulence to determine the turbulent macroscale. Operationally, however, after some terms are modelled, the dissipation transport equation is a special case of a more general length scale equation ...*

Rodi [1987]:

> *The arguments for the relative merits of the ε and the kL equations are rather academic because both equations are fairly empirical and, with the constants suitable adjusted, perform in a similar manner. One difference is that the kL equation requires an additional nearwall term ... while the kL equation does not.*

It is beyond the scope of this report to be more in favour of either or the other argument. However, on a visit to the Joint Research Centre in April 1998, George Mellor stated that the $k - \varepsilon$ model, as it is implemented in GOTM, is a MellorYamada-model as well. He said this because we adopt here most of the derivations motivated in Mellor and Yamada [1974, 1982], and (besides using other stability functions) only substitute the length scale equation suggested by them by an equation for the dissipation rate ε .

26.2.1.1 The ε equation

Similarly to the TKE-equation (2.38), an exact transport equation for the dissipation rate ε of TKE can be derived from the Navier-Stokes equations. However, a mathematical closure of this equation is only achieved after a number of simplifying model assumptions with the consequence that three new empirical parameters have to be introduced:

$$\partial_t \varepsilon - \partial_z (\upsilon_\varepsilon \partial_z \varepsilon) = \frac{\varepsilon}{\kappa}(c_{\varepsilon 1} P + c_{\varepsilon 3} B - c_{\varepsilon 2} \varepsilon) . \qquad (103)$$

By means of equation (2.34), the macro length scale L can now be calculated. The eddy diffusivity for " in equation (2.46) is modelled as:

$$\upsilon_\varepsilon = \frac{\upsilon_t}{\sigma_\varepsilon} \qquad (104)$$

with $\sigma_\varepsilon = 1.08$ (see equation (3.18) and Burchard et al. [1998]).

The values $c_{\varepsilon 1} = 1.44$ and $c_{\varepsilon 2} = 1.92$ are adopted from Rodi [1980]. It should be noted that the buoyancy production related parameter $c_{\varepsilon 3}$ in the ε equation used here deviates from the standard literature (see e.g. Rodi [1987]) where this value is non-negative. The demand for a negative $c_{\varepsilon 3}$ has recently been shown by Burchard and Baumert [1995] and Burchard et al. [1998], but the actual value is sensitive to the stability function chosen. Burchard et al. [1998] calibrated $c_{\varepsilon 3}$ by means of a numerical experiment similar to the KatoPhillips experiment (see section 6.1.2). When using the Galperin et al. [1988] stability functions, $c_{\varepsilon 3} = -0.4$ gave the best agreement between theoretically derived and simulated entrainment velocity.

A negative value for $c_{\varepsilon 3}$ provides a source for dissipation in the case of stable stratification. This has a similar effect than the lower limit for ε, equation (2.72), which is included into the model in order to parameterize the effect of transformation of turbulent eddies into internal waves. For unstable stratification, the classical value $c_{\varepsilon 3} = 1.0$ is used here.

By combining (2.34) and (2.45), a Dirichlet boundary condition for ε can be derived,

$$\varepsilon = (c_\mu^0)^3 \frac{k^{3/2}}{k(\tilde{z} + z_0)} . \qquad (105)$$

By differentiating (2.48) with respect to $\tilde{z}$ and considering (2.41) and (2.42), an equivalent flux boundary condition for ε can be achieved [2] :

$$\frac{\upsilon_t}{\sigma_\varepsilon}\partial_{\tilde{z}}\varepsilon = -(c_\mu^0)^3 \frac{\upsilon_t}{\sigma_\varepsilon}\frac{k^{3/2}}{k(\tilde{z} + z_0)^2} \qquad (106)$$

where $\tilde{z}$ is again the distance from bottom or surface, respectively. The reasons for considering a flux condition for ε are the numerical problems caused by high gradients near the boundaries (see Stelling [1995], Burchard and Petersen [1999]). The different numerical performance of ε boundary conditions (2.48) and (2.49) in boundary layer flow will be discussed in section 3.5.

26.2.1.2 The kL equation

Even more than the ε equation (2.46), the kL equation which has been designed by Mellor and Yamada [1982] is a result of many empirical considerations. After adapting the notations to those used here in this report, the kL equation reads as:

$$\partial_t(kL) - \partial_z(\upsilon_L \partial_z(kL)) = L\left(c_{L1}P + c_{L3}B - \left(1 + E_2\left(\frac{L}{L_z}\right)^2\right)\varepsilon\right). \tag{107}$$

The empirical parameters in this equations are $c_{L1} = c_{L3} = 0.9 \; and \; E_2 = 1.33$. In (2.50), a wall proximity function with a barotropic length scale L_z has to be prescribed in order to guarantee the loglaw near boundaries, see sections 3.2.1 and 3.2.2 and Mellor and Yamada [1982]. Two possible profiles which are both included in GOTM have been tested by Burchard et al. [1998], a parabola shaped one:

$$L_z = k\frac{\left(d_b + z_0^b\right)\left(d_s + z_0^s\right)}{\left(d_b + z_0^b\right) + \left(d_s + z_0^s\right)}, \tag{108}$$

and a triangle shaped one:

$$L_z = k\min\left(d_b + z_0^b, ds + z_0^b\right). \tag{109}$$

Here, d_s is the distance from the surface and d_b the distance from the bottom.

As boundary values for L, the condition (2.45) is used:

$$L = k\left(d_b + z_0^b\right), \tag{110}$$

$$L = k\left(d_s + z_0^s\right), \tag{111}$$

26.2.2 Algebraic forms

Instead of solving a transport equation to obtain the turbulent mixing length, it is possible to calculate it from a simple geometric or a more complex diagnostic expression.

26.2.2.1 Simple geometric forms

Geometric expressions are designed in a way that they fulfill the logarithmic law of the wall (2.45) near boundaries. Two simple expressions that satisfy this requirement in the presence of surface and bottom boundaries are the parabolic

$$L = k\frac{\left(d_b + z_0^b\right)\left(d_s + z_0^s\right)}{\left(d_b + z_0^b\right) + \left(d_s + z_0^s\right)} \tag{112}$$

and the triangle profile

$$L = k\min\left(d_b + z_0^b, d_s + z_0^s\right), \tag{113}$$

where d_b and d_s are the distances to bottom and surface, respectively, and z_0^b and z_0^s are the roughness lengths of bottom and surface boundaries, respectively (see (2.1) and (2.2)). Other expressions take into account that the position of the maximum in mixing length may not lay in the middle of the water column, as observed to occur in tidal flows. This anisotropy in the profile is obtained by changing the functional form of surface or bottom mixing length, as suggested by Robert and Ouellet [1987] (note that this function is a distorted parabola)

$$L = k\left(d_b + z_0^b\right)\left(1 - \frac{d_b}{D}\right)^{1/2} \tag{114}$$

or Xing and Davies [1995]

$$L = k\left(d_b + z_0^b\right) e^{-\beta d_b/D}, \tag{115}$$

where β is a tuning parameter. For (2.57) and (2.58), the logarithmic law of the wall (2.45) is not any more fulfilled for the surface. This is in order to take into account free surface effects like breaking waves. Other authors suggest that the turbulent mixing length should depend on local characteristics of the turbulent flow like TKE. The most widespread of these methods is that of Blackadar [1962]

$$L_\alpha = \left(\frac{1}{k\left(d_b + z_0^b\right)} + \frac{1}{L_0}\right)^{-1} \tag{116}$$

with L_0 given by the expression:

$$L_0 = \gamma_0 \frac{\int_{-H}^{\zeta} k^{1/2} z dz}{\int_{-H}^{\zeta} k^{1/2} dz}, \tag{117}$$

where γ_0 is a constant in the range from 0.1 to 0.4. In the presence of two boundaries, the Blackadar mixing length can be written in a form that automatically gives an approximation to a linear profile of L near walls:

$$L = \left(\frac{1}{k\left(d_s + z_0^s\right)} + \frac{1}{L_\alpha}\right)^{-1}. \tag{118}$$

with L_α from (2.59).

For stratified flow, the mixing length is also affected by stratification. A way of including stratification into these simple geometric expressions for the mixing length is multiplying it by a stratification correction factor,

$$L \rightarrow LF_s. \tag{119}$$

Some proposed forms for this factor are based on measurements of the atmospheric boundary layer, Geernaert [1990]:

$$F_s = \begin{cases} 1 - \alpha R_i, & for\ R_i \succ 0 \\ \left(1 - \beta R_i\right)^{1/4} & ,else \end{cases} \tag{120}$$

with $R_i = N^2/M^2$, the gradient Richardson number and the parameters α (which can range from 5 to 10) and β = 15.

Nihoul and Djenidi [1987] have suggested

$$F_s = 1 - R_f \tag{121}$$

with the flux Richardson number $R_f = -B/P$. Stability functions (introduced in section (2.2.3)) have a similar effect as this correction factor: they increase mixing for unstable stratification and decrease it for stable stratification. This makes it difficult to calibrate a model that uses a stratification-depending functional form of both. Moreover, a calibration of the model with a given mixing length corrected by a stratification factor may no longer be valid if stability functions are changed.

26.2.2.2 Complex forms

A more complex diagnostic mixing length is the one proposed by Therry and Lacarr`ere [1983] and adapted for GOTM from Blanke and Delecleuse [1993]. Two characteristic lengths, one for dissipation, L_ε, and another for mixing, L_k, are defined as combinations of two master length scales L_u and L_d obtained in each point as the upward or downward distance that a fluid particle placed at that point must travel to convert TKE in potential energy:

$$\begin{aligned} &\int_z^{z+L_u(z)} \left(b(z) - b(\xi)\, d\xi \right) = k(z) \\ &\int_{z-L_d(z)}^{z} \left(b(\xi) - b(z) \right) d\xi = k(z). \end{aligned} \tag{122}$$

L_k and L_ε are then calculated as

$$\begin{aligned} &L_k(z) = \min\left(L_d(z), L_u(z) \right) \\ &L_\varepsilon(z) = \left(L_d(z)\, L_u(z) \right)^{1/2}. \end{aligned} \tag{123}$$

The expressions for L_k and L_ε are designed such that they take into account the fact that a strongly stratified region affects diffusion as a wall does, while dissipative phenomena, even near a wall, are affected by the maximum vertical scale of convective motions. Deviating from the usual structure of GOTM, in this model, the dissipation rate (2.34) and the eddy viscosity (2.31) and diffusivity (2.32) are calculated with different length scales:

$$L = \begin{cases} \dfrac{c_\mu}{\tilde{c}_\mu} L_k & \text{for insertion into (2.31),} \\[2ex] \dfrac{c_\mu'}{\tilde{c}_\mu} L_k & \text{for insertion into (2.32),} \\[2ex] \dfrac{c_\varepsilon}{\left(c_\mu^0\right)^3} L_\varepsilon & \text{forinsertion into (2.34),} \end{cases} \tag{124}$$

where $c_\varepsilon = 0.7$ and $\tilde{c}_\mu = 0.1$ are constants (see Gaspar et al. [1990]).

In the algebraic mixing length parameterization used in the ISPRAMIX ocean circulation model (see Eifler and Schrimpf [1992] and Demirov et al. [1998]), three different regions are distinguished:

the top and the bottom mixed layers and a stably stratified interior layer. After calculating the height h_m of each mixed layer (obtained as the first point from the boundary with $k < k_{\min} = 10^{-5} Wkg^{-1}$),the macro length scale L in both mixed layers is obtained from a Blackadar [1962] type formula:

$$L = \frac{k\tilde{z}}{1 + \dfrac{k\tilde{z}}{c_2 \cdot h_m}} \left(1 - R_f\right)^e \tag{125}$$

where $\tilde{z}$ is the distance from the interface (surface or bottom). Equation (2.68) predicts an approximation to a linear behaviour of L near boundaries (see eq. (2.45)) and a value proportional to the thickness of the mixed layer far from the interface, $L = c_2 h_m$, where

$c_2 = 0.065$ is estimated from experimental data as discussed in Eifler and Schrimpf [1992]. The factor $\left(1 - R_f\right)$, with the flux Richardson number $R_f = -B/P$, accounts for the effect of stratification on the length scale. The parameter e is here a tuning parameter (pers. comm. Walter Eifler, JRC, Ispra, Italy) which is usually set to e = 1. However, during the simulations of the KatoPhillips experiment, we found that a higher value was necessary in order to predict the expected mixed layer depth evolution (see section 6.1.2).

Below the top mixed layer and above the bottom friction layer, a suggestion of Zilitinkevich and Mironov [1992] is applied to calculate the length scale in the interface L i between the thermocline and the mixed layer. This formula parameterizes the penetration of the mixed layer into the stably stratified thermocline by turbulence diffusion:

$$L_i = c_i \frac{k^{1/2}}{N}. \tag{126}$$

The constant c_i is determined for every specific case (and for both boundary layers) by making use of the requirement that the length scale in the mixed layer L_m and near the thermocline must match at $z = h_m$. Therefore, we have a matching condition c_i to determine c i for each boundary layer:

$$c_i = \left(\frac{L_m N}{k^{1/2}}\right)_{z=h_m}. \tag{127}$$

In the core region of the thermocline, a constant value of L typical for stratified conditions is imposed, where (2.70) would yield too small values. In GOTM, we use

$L = 0.01m$. Care has been taken that the model behaves well in every possible situation. Thus, if the two boundary layers overlap, we suppose there is no stably stratified region, only two boundary layers, with L calculated from (2.68). The same is supposed if only one mixed layer is detected. This way, we guarantee that the length scale satisfies the linear profile near both boundaries.

26.2.3 Length scale limitation

In order to include the limiting effect of stable stratification, Galperin et al. [1988] found it necessary to introduce an upper limit for the macro length scale L in stably stratified flows,

$$L^2 \leq \frac{0.56k}{N^2} \quad for\ N^2 > 0. \tag{128}$$

Applied to the ε equation, this corresponds to a lower limit for the dissipation rate ε , which can be calculated by means of using (2.45) as

$$\varepsilon^2 \geq 0.045k^2N^2 \quad for\ N^2 > 0. \tag{129}$$

In section 6.1.2, the effect of these limitations for a simple mixing experiment is shown.

26.3 Stability functions

In the literature many different sets of stability functions c_μ and $c_\mu^{'}$ are found (see equations 2.31 and 2.32). They have the function to increase mixing for instable and decrease mixing for stable stratification.

2.2.3.1 Simple stability functions

The simplest set of stability function considers the stability function for momentum, c_μ , as a constant. The stability function for tracers, $c_\mu^{'}$, is related to c_μ via the Prandtl number P_r :

$$c_\mu = c_\mu^0 = const \ \ (see\ \ (2.34)) \tag{130}$$

$$c_\mu^{'} = \frac{c_\mu}{P_r}. \tag{131}$$

For the very simple case, the Prandtl number is a constant of the order of unity.

A gradient Richardson number depending approach had been suggested by Munk and Anderson [1948]:

$$P_r = \begin{cases} P_r^0 \dfrac{(1+3.33R_i)^{3/2}}{(1+10R_i)^{1/2}}, & for\ R_i \geq 0, \\ P_r^0\ , & for\ R_i < 0 \end{cases} \tag{132}$$

with the constant Prandtl number for neutral stratification, P_r^0 .

In the ISPRAMIX ocean model (see section 2.2.2.2.1), another approach is used for considering stability effects on vertical mixing. The stability functions in this model are of the form:

$$c_\mu = const = 0.5, \tag{133}$$

$$c_\mu^{'} = c_\mu f\left(R_f\right) = c_\mu \frac{1}{P_r^0}\left(1 - R_f\right)^{1/2}. \tag{134}$$

The neutral Prandtl number used there is $P_r^0 = 0.7143$. The function $f\left(R_f\right)$ is assumed to lay between the values 0.18 (corresponding to a supercritically stratified situation) and 2.0 (preventing it from growing too much under unstable conditions).

A formulation for $\left(1 - R_f\right)$ can be derived from the definition of the flux Richardson number

$$R_f = \frac{c_\mu^{'}}{c_\mu} R_i \tag{135}$$

and (2.77) (see Beckers [1995]):

$$\left(1 - R_f\right) = \left[\left(\tilde{R}_i^2 + 1\right)^{1/2} - \tilde{R}_i\right]^2 \tag{136}$$

with

$$\tilde{R}_i = \frac{0.5}{P_r} R_i \tag{137}$$

where R_i is the gradient Richardson number.

26.3.1 Mellor and Yamada [1974]

Mellor and Yamada [1974] derived stability functions from prognostic equations for Reynolds stresses and heat fluxes under the assumption of a local equilibrium of stresses and heat fluxes.

They obtained the following system of equations for c_μ and $c_\mu^{'}$ which is for simplicity here given with coefficients readily calculated from the empirical parameters used:

$$\begin{aligned} 2.0424\alpha_M \, c_\mu + \left(1 + 15.2958\alpha_N\right) c_\mu^{'} &= 1.0465 \\ \left(1 + 2.5392\alpha_M + 3.0636\alpha_N\right) c_\mu + 8.1142\alpha_N \, c_\mu^{'} &= 0.9888. \end{aligned} \tag{138}$$

with the nondimensional shear

$$\alpha_M = \frac{L^2}{k} M^2. \tag{139}$$

and the nondimensional buoyancy parameter

$$\alpha_N = \frac{L^2}{k} N^2. \tag{140}$$

With the constraints

$$\alpha_N \geq -0.064 \tag{141}$$

and

$$\alpha_M \leq 1.65 + 25\alpha_N, \tag{142}$$

the solutions for c_μ and $c_\mu^{'}$ are unique and positive (see figure 2.2). If one of the constraints (2.84) or (2.85) is violated, α_M or α_N will be set to the threshold value and as such inserted into (2.81).

This might lead to numerical instabilities for unstable stratification.

26.3.2 Galperin et al. [1988]

For the local equilibrium $P + B = \varepsilon$, the following relation between nondimensional parameters can be directly obtained:

$$c_\mu \alpha_M - c_\mu^{'} \alpha_N = \left(c_\mu^0\right)^3. \tag{143}$$

If this is inserted into the exact form of (2.81), then the so called quasiequilibrium stability functions are obtained, which only depend on the nondimensional buoyancy parameter α_N:

$$c_\mu = \frac{c_\mu^0 + 2.182\alpha_N}{1 + 20.40\alpha_N + 53.12\alpha_N^2} \tag{144}$$

and

$$c_\mu^{'} = \frac{0.6985}{1 + 17.34\alpha_N}. \tag{145}$$

Also here, we display the equations with the empirical parameters already inserted. These stability functions are commonly referred to as the Galperin et al. [1988] the stability functions.

[FIND Figure]

Figure 23. Stability functions (left) and (right) according to Mellor and Yamada [1974].

The constraint for α_N given by Galperin et al. [1988] is:

$$\alpha_N > \alpha_{\min} = -0.0466. \tag{146}$$

In the case of convection, much smaller values of α_N may occur. In order to guarantee a smooth transition into this convective regime, α_N is here limited by

$$\alpha_N = \max\left\{\tilde{\alpha}_N, \tilde{\alpha}_N - \left(\tilde{\alpha}_N - \alpha_c\right)^2 / \left(\tilde{\alpha}_N + \alpha_{\min} - 2\alpha_c\right)\right\} \tag{147}$$

with $\tilde{\alpha}_N = L^2 N^2 / k^2$ (see equation (2.83)) and $\alpha_c > \alpha_{\min}$ min (see Burchard and Petersen [1999]). For free convection simulations, it was sufficient to use $\alpha_c = -0.02$. An upper limit for α_N is given by $\alpha_{\max} = 0.56$ (see Galperin et al. [1988]). The stability functions c_μ and $c_\mu^{'}$ are shown in figure 2.3 for different values of α_c.

26.4 Shear instability and IW parameterization

A major weakness of local turbulence models is that they do not take into account the increasing effect of mixing that shear instability and internal wave activity induce in the presence of stable stratification. Some simple methods to model this effect in the framework of a secondmoment closure model have been introduced in GOTM.

26.4.1 Limitation of turbulent magnitudes

The first obvious way of taking into account stable stratification effects is to impose a limitation on turbulent magnitudes by prescribing an upper limit of the turbulent length scale (or equivalently, a lower limit of the dissipation rate ε). The conditions (2.71), suggested by Galperin et al. [1988]

Figure 24. Quasiequilibrium stability functions Error! Objects cannot be created from editing field codes. and Error! Objects cannot be created from editing field codes. according to Galperin et al. [1988], but smoothed for unstable stratification. Curves for different critical stratifications are shown (see also eq. (2.90)).
Error! Objects cannot be created from editing field codes. unsmoothed (d). This figure has been taken from Burchard and Petersen [1999].

and Luyten et. al. [1996a], are optionally used in GOTM. Luyten et al. [1996b] found, that this limitation only has a noticeable effect on results, if a lower limit for the turbulent kinetic energy is set. The oceanic observations by Gregg [1987] suggest that it is reasonable to assume that turbulent kinetic energy tends to a constant limiting value under stably stratified conditions. The actual value of this constant is obtained by comparing measurements and model predictions, and can act as a tuning parameter of the model as illustrated by Burchard et al. [1998].

26.4.2 Mellor [1989]

Mellor [1989] suggests to introduce internal wave mixing as an extra term in the turbulence equations, instead of imposing a limiting condition. The basic idea of the model is that internal waves induce an additional shear that is added to the mean flow shear. As internal wave energy is known to depend on, the internal wave shear is modelled as a term αN_2, which is added to the shear terms (2.35) in both turbulent transport equations. Instead of (2.35), for stable stratification

$$P = \upsilon_t \left(M^2 + \alpha N^2 \right) \tag{148}$$

is used.

After estimating the difference between observed shear profiles and modelled ones, a value of $\alpha = 0.7$ is suggested.

PART III – APPENDICES

27 PARAMETERS IN EIA 3D MODEL

27.1 Explanatory abbreviations (in run control file)

HS:	Depth at velocity point
HZ:	Depth at water level point
HSO:	Bottom level at water level point
UV:	Water velocity component at East and North direction
Z:	The initial water level elevation
ICET:	index
ICEZ:	index
ZL:	Layer elevation
TUR:	Turbulence
KINE:	Kinetic Energy
EPS:	Dissipation rate or Kinetic energy
TD WAVES/C/:	Wave energy
U:	East direction velocity component
V:	North direction velocity component
W:	Upward velocity
UA:	East direction velocity average
VA:	North direction velocity average
DIR:	Flow direction
SPE:	Flow speed
FLX:	East direction flux
FLY:	North direction flux
FLZ:	Upward flux

27.2 Output selection symbols (in GUI and graphics files)

TEMP	Temperature (C)
SALI	Salinity (mg/l)
XXXX	Other concentration or density variables named XXXX
SURF	Water surface displacement from equilibrium or flooding water depth in a concentration point (cm)
ZLEV	Isopycnal layer interface elevation (cm) (at concentration grid point)
ZPRE	Isopycnal layer interface pressure ( ) (at concentration grid point)
DEPS	Depth at flow velocity grid point (cm)

DEPZ	Depth at concentration grid point (cm)
ELES	Water surface elevation at flow velocity grid point (m)
ELEZ	Water surface elevation at concentration grid point (m)
XVEL	Layer flow component towards grid east (cm/s)
YVEL	Layer flow component towards grid north (cm/s)
XVEA	Total depth averaged flow component towards grid east (cm/s)
YVEA	Total depth averaged flow component towards grid north (cm/s)
VELL	Layer flow velocity (cm/s) (=SPED)
VELA	Total depth average flow velocity (cm/s)
SPED	Layer flow speed (cm/s)
DIRE	Layer flow direction (degrees clockwise from grid north)
DIRA	Total depth average flow direction (degrees clockwise from grid north)
DIRE	Layer flow direction (degrees clockwise from grid north)
SMAG	Calculated horizontal viscosity (Smagorinsky) (cm2/s)
ZVIS	Vertical viscosity (cm2/s)
ZKIN	Turbulent vertical kinetic energy (cm2/s2)
ZEPS	Dissipation of turbulent vertical kinetic energy (cm2/s3)
FETC	Fetch (m)
FETS	Fetch (wind stress) ratio (fetch/maximum fetch)
WHEI	Wave height (cm)
OBVE	Orbital bottom velocity (cm/s)
OBST	Wave generated bottom stress (cm/s)
OPER	Wave period (1/s)
VEGC	Vegetation coverage (%)
ICES	Ice coverage (%)
ICEZ	Ice thickness (cm)
ICET	Ice temperature (C)
DIKE	Dike on/off

27.3 Parameters used in the field drawing

GRID	Model Grid
WQGRID	Water Quality Grid
F_GRELEV	Ground Elevation
F_MAXH	Maximum Depth
F_AVH	Average Depth
F_LU	Land Used Class
F_MAXT	Peak Flood Time

F_DURT	Flood Duration Time
F_ARRT	Flood Arrival Time
F_DRYT	Flood Drying Time
F_SPSAV	Speed Surface Average
F_SPMAV	Speed Middle Average
F_SPBAV	Speed Bottom Average
F_SPVAV	Speed Vertical Average
F_CSA_SEDI	Sediment Surface Average
F_CMA_SEDI	Sediment Middle Average
F_CBA_SEDI	Sediment Bottom Average
F_CVA_SEDI	Sediment Vertical Average
F_CVMAX_SEDI	Sediment Maximum Value
F_CVMIN_SEDI	Sediment Minimum Value
F_BOTSEDI	Net Sedimentation at the End
F_BOTAVSEDI	Average Net Sedimentation at End
F_CSA_OXYG	Oxygen Surface Average
F_CMA_ OXYG	Oxygen Middle Average
F_CBA_ OXYG	Oxygen Bottom Average
F_CVA_ OXYG	Oxygen Vertical Average
F_CVMAX_ OXYG	Oxygen Maximum Value
F_CVMIN_ OXYG	Oxygen Minimum Value

etc.

28 SHORT DESCRIPTION OF GRID GENERATION ALGORITHM

The grid generation starts from the digital depth point and line data. The grid area is defined as well as final resolution. The depth grid is first calculated to much dense grid, typically there is 5x5 small depth cells in one final grid cell. It is possible to get more accurate final grid this way. First, given depths are set to the dense grid.

The basic idea is the inverse distance weight of nearest depth points. The distance from the nearest depth point is calculated by stepping one cell away. Stepping stops, if stepping from other depth point will be encountered or depth contour line will be encountered. The advantage of the stepping method is that points do not affect over other points, but affect also in the complex canals (Figure 25).

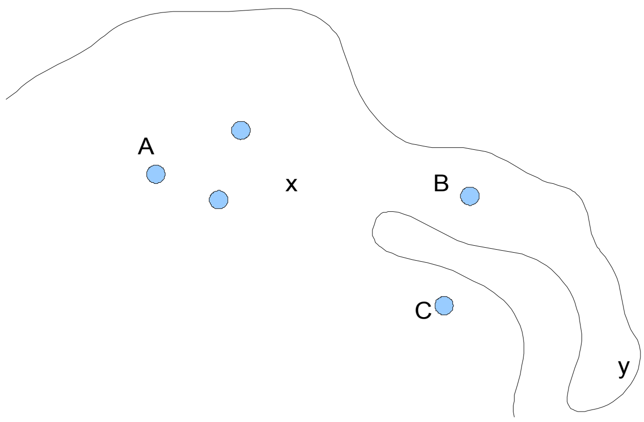


Figure 25. Depth point B affect to the point x, but point A not. B also affect to the point y, but C not.

Similar stepping is calculated from depth lines. If distance from two different depth lines is much smaller (given constant) than any depth points, the depth are calculated linearly from these, otherwise, depth points are also used. Depth lines are not used, if the nearest points are depth points.

First new depths are calculated to the areas, where only depth lines affect. Then, loop starts from the longest distance from all depth points. The inverse distance weighted depth is set, if three or more steppings encountered at the point or point is next to the depth line or depth is set at the earlier round.

This algorithm leaves empty parallelogram shaped areas. The depth at the center of parallelogram is calculated from its' border values. The parallelogram is filled linearly from the central depth and nearest border value. The advantage, compared to the triangular method, is that the saddle like depth distribution is allowed inside the parallelogram.

Final grid is calculated by combining cells. If more than 50% of the final cell is water, the depth is the average of water cells and otherwise depth is zero.