SC001/98

DUFLOW - Manual

EDS

Leidschendam

International Institute for
Infrastructural, Hydraulic and
Environmental Engineering

The Netherlands

DUFLOW - Manual

DUFLOW

A micro-computer package for the simulation of one-dimensional unsteady flow and water quality in open channel systems

Manual	Software
First edition, June 1989 Delft, the Netherlands	duflow version 1.00
Edition 2.0, November 1992 Wageningen, The Netherlands	duflow version 2.00 - 2.03
Edition 2.1, December 1995 Leidschendam, The Netherlands	duflow version 2.04

Contributing organizations

1. International Institute for Hydraulic and Environmental Engineering, Oude Delft 95, Delft, The Netherlands
2. Rijkswaterstaat, Tidal Waters Division, Koningskade 4, The Hague, The Netherlands
3. Delft University of Technology, Faculty of Civil Engineering, Stevinweg 1, Delft, The Netherlands
4. Agricultural University of Wageningen, Department of Nature Conservation, Ritzema Bosweg 32a, Wageningen, The Netherlands
5. Stichting Toegepast Onderzoek Waterbeheer (stowa), Arthur van Schendelstraat 816, P.O. Box 8090, 3503 RB Utrecht.

Distribution and information

EDS
Loire 206
P.O. Box 406
2260 AK Leidschendam
The Netherlands

Phone (31-70) 30 14 600
fax (31-70) 32 07 999
telex 32683 nl.

Contents

1. Introduction

1.1 History and purpose of the model

The introduction of personal or micro computers in organisations involved in water management and hydraulic engineering stimulated the demand for easy-to-use computer models.
Water authorities require a dynamic management of their extensive water systems and related infrastructure to provide water for industry, agriculture, domestic supply, fishery, energy and water quality control, taking into account reduction of damage due to excess of water, etc. In hydraulic engineering a proper design and operation of river based structures and improvement works requires also consideration of larger parts of the overall water system.

The use of real-world models suiting a wide range of users and their applications has become a prerequisite for optimal design and management.

Various educational institutions stressed on having free access to a hydrodynamic user-oriented computer code. The Department of Sanitary Engineering and Water Management, Faculty of Civil Engineering, Delft University of Technology and the International Institute for Hydraulic and Environmental Engineering, Delft, stimulated various organisations to contribute to this model development.

Three institutes in The Netherlands, viz.

- The International Institute for Hydraulic and Environmental Engineering (IHE), Delft;
- The Rijkswaterstaat (Public Works Department), Tidal Waters Division, The Hague;
- The Delft University of Technology, Faculty of Civil Engineering,

thus collaborated in providing the engineering community with a PC-program package for unsteady flow computations in networks of open water courses. The computational core of this model is based on the FORTRAN computer code IMPLIC which is originally developed by the Rijkswaterstaat. To improve its user-orientation this implic code has been fitted with a user interface written in ms-Basic. At the same time computational procedures were updated and more options for structures were included.

In 1992 version 2.0 was completed. In this version the package was extended with water quality modelling. The water quality part was developed by the Agricultural University of Wageningen / Department of Nature Conservation in co-operation with the International Institute for Hydraulic and Environmental Engineering (ihe). Especially as the relationship between quality and flow gets special attention nowadays, a package suitable for modelling both aspects makes DUFLOW a useful tool in Water Quality Management.

In the Water Quality part the process descriptions can be supplied by the user. This special concept enables the user to create different types of water quality models. Two predefined eutrophication models are included in the package.

The package as a whole is called du(tch)flow. It is intended to serve both practising engineers and students and is available at nominal costs.

Access to the source code is only allowed to those institutions who will contribute a substantial extension to the model package.

A free student's version is available, which includes all options, but is restricted in the number of channel sections and structures. Information on purchase and users rights will be supplied by EDS, Leidschendam, The Netherlands.

This chapter gives a general overview of the capabilities of the model and is intended for those who have to decide whether to use duflow or not.

Chapter 2 gives an account of the physical and mathematical background; it provides advanced users with the necessary information on the assumptions and methodologies underlying the model.

Chapter 3 describes the use of the model; it can be consulted as a reference guide, although the more experienced users may find enough information in the additional screen messages and menus. Those who are using duflow for the first time are referred to appendix B (Getting started) where a worked out example is presented step by step.

Chapter 4 describes a number of (simplified) real-world problems in various areas of application; on one hand this chapter is also part of the scientific account, on the other hand it is helpful to beginning users since it shows how to solve several types of engineering problems.

Chapter 5 gives conclusions and suggestions for future extensions.

1.2 Types of users

The duflow program package is designed for various categories of users. The model can be used by water managers and designers and has proved to be a very useful tool in education. Since it runs on IBM-compatible micro-computers it can be operated in almost every scientific or engineering environment.

In engineering education the major advantage is the short learning time due to its menu structure and screen-oriented input and output.

In water management the model can be used to simulate the behaviour of a system due to operational measure such as opening or closing of sluices, switching on pumping stations, reduction of pollutant loads etc. and thus to optimise the day-to-day management decisions and to evaluate management strategies.

In a consultancy environment the model can be used in the design of hydraulic structures, flood and salt prevention and river training measures.

1.3 Design considerations

duflow is designed to cover a large range of applications, such as propagation of tidal waves in estuaries, flood waves in rivers, operation of irrigation and drainage systems, etc.

Basically free flow in open channel systems is simulated, where control structures like weirs, pumps, culverts and siphons can be included.

As in many water management problems the runoff from catchment areas is important, a simple rainfall-runoff relation is part of the model set-up. The selected numerical scheme allows for a rather large time step in the computation and for choosing different lengths of the elementary sections.

Water quality is an increasing concern in water management, e.g. problems of algal bloom, contaminated silt, salt intrusion etc. More often than not water quality has to be described by a (sometimes large) number of parameters, so

duflow allows for a number of quality constituents, and it is able to model the interactions between these constituents. There is an abundance of formulations around, so there is a great freedom for the user in formulating the production or destruction of biological or chemical materials. To test such a formulation there is the possibility of a simple box model. For common users a number of pre-formulated interactions is available where still the coefficients can be influenced.

An important topic in water quality problems is also the interaction between the bottom layer and the water mass above; duflow distinguishes transported materials that flow with the water and bottom materials that are not transported but that can be subject to similar interactions as described above.

duflow can direct results to disk files which can be approached by other computer programs (ground water flow, economic analysis and structural design).

For immediate inspection, the results can be graphically displayed on the screen in time or space. Optionally output is given in the form of tables, while all output can be directed to the (graphical) printer.

duflow is efficient both in terms of computation time and required memory, thus allowing the processing of large models. Computation time is usually in the range of minutes up to one hour.

1.4 Options and elements

In the duflow system a model, representing a specific application, can be put together from a range of elements. Types of elements which are available are open channel sections (both river and canal sections), and control sections or structures such as weirs, culverts, siphons, and pumps.

For instance an irrigation or drainage system consists of a network of (small) canals; water may be locally transported through pumps and siphons, and in the network the discharges and levels may be controlled by means of weirs.

In case of a flood wave in a river the discharge imposed at the upstream boundary of a river stretch is transmitted through a sequence of river sections which may be separated by (movable) weirs.

Boundary conditions can be specified as:

- water levels and discharges, either constant or in the form of time series or Fourier series;
- additional or external flow into the network can be specified as a (time dependent) discharge or can be computed from a given rainfall, using a simple rainfall-runoff relation;
- discharge-level relations (rating curves) in tabular form;
- concentrations and loads of all the transported materials in the quality model, either constant or in the form of time series or Fourier series.

Wind stress which may be significant in extensive shallow networks can also be included. All time series can directly be defined or can be read from an external ASCII file.

A bird's-eye view of the network provided by the program allows a quick check of the network including its orientation and the connections between the nodes. Simple shaped cross-sections can be specified with only a few data. Two types of resistance formulae can be used, viz. Manning and De Chézy.

For more complex cross-sections (natural rivers) the width (both width of flow and width of storage) and the friction factor and the hydraulic radius can be specified as a function of the water level.

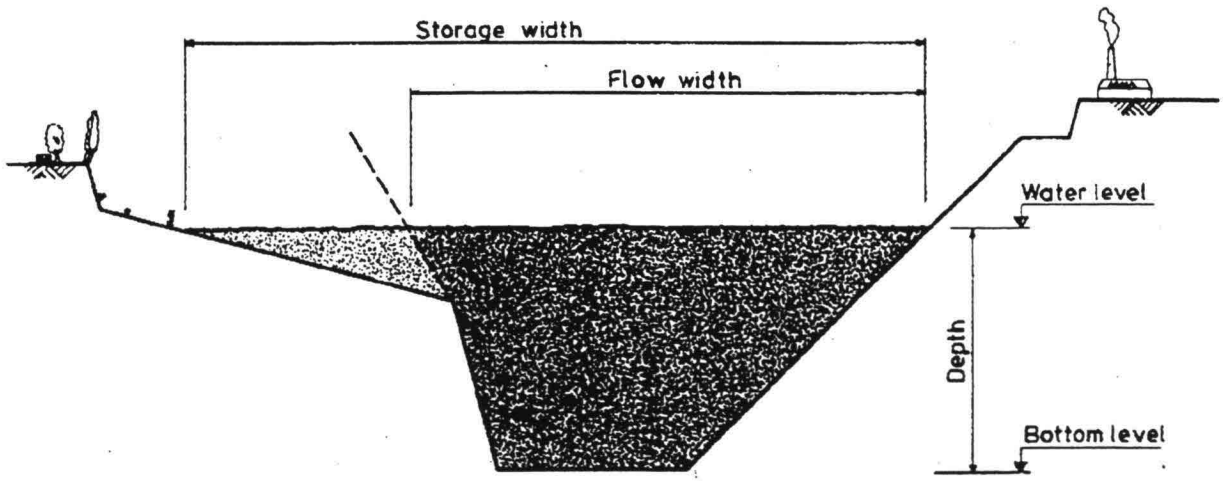


Figure 1.1 Cross sectional distribution

A range of control structures is available in duflow:

Weirs with movable gates can be modelled for overflow and underflow (submerged) conditions. Transition between various flow situations such as overflow and underflow, sub- and supercritical flow in either direction takes place automatically.

During execution of the program the height and width of a gate can be modified depending on the actual computed water level(s) at a pre-defined location in the system and according to a pre-specified procedure (trigger conditions). Thus various strategies of 'automatic' gate manipulation can be specified.

A Culvert is represented as a weir by taking into account also the friction. Pumps are automatically operated depending on the upstream level only.

Siphons are defined as circular pipes and the flow depends on the water levels at both sides. Friction losses are taken into account.

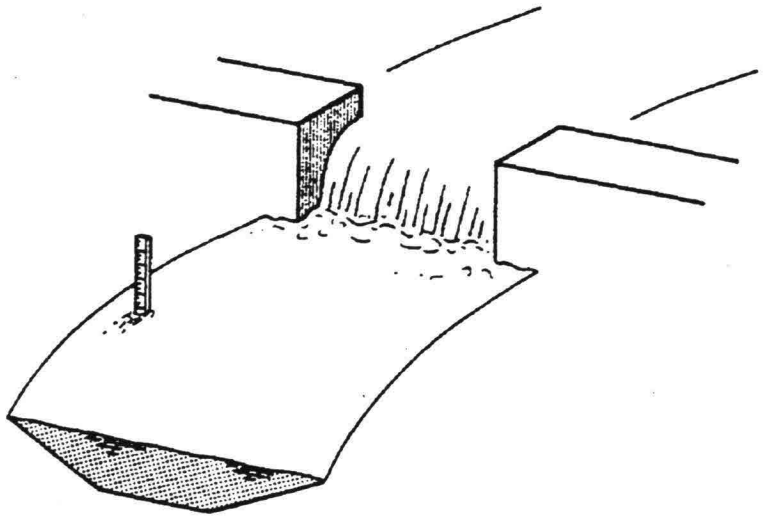


Figure 1.2 Sketch of a hydraulic structure

1.5 Structure of the duflow package

The duflow system consists of three modules which are controlled by the master menu. The interactive input module (pre-processor) and output module (post-processor) are written in ms-Basic, the non-interactive computational module is written in Fortran-77.

The data prepared by the input module are stored in the network and boundary files, which contain all information on computation, network characteristics and hydraulic conditions. Results of the computation are written to the result file and can be used for further processing.

Flow and quality can be simulated separately or simultaneously. Using the first option the necessary hydraulic variables are stored on disk and read during simulation of quality. Another option available enables to simulate the processes only. In that case transport is eliminated.

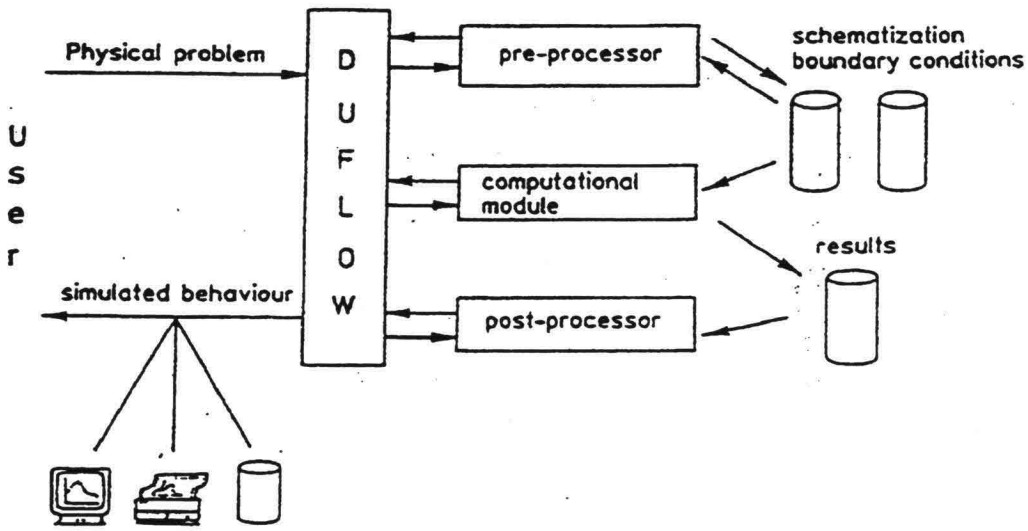


Figure 1.3 DUFLOW package

1.6 Software and computer specifications

The minimum required computer configuration is an ibm-compatible PC (XT, AT or 386-class) with 640 Kbytes internal memory, a graphical card (CGA, EGA, VGA, OLIVETTI OR HERCULES graphics), a hard disk and preferably a mathematical co-processor.

For special applications of duflow the computational module can be made available on other computers than PC, for example SUN, HP9000 or VAX.

2. Physical and Mathematical background

2.1 Introduction

In this chapter an outline is given of the basic equations used in DUFLOW and the numerical procedures used to discretize and solve these equations.

Paragraph 2.2 deals with the equations for unsteady flow in open channels. In special paragraphs the use of boundary-, initial conditions and structures is presented.

In paragraph 2.3 the mass transport equation and numerical solution applied in the quality part are discussed. Special attention is paid to the use of boundary conditions and process descriptions.

Paragraph 2.4 gives an outline of the numerical methods used. Finally in paragraph 2.5 some practical considerations for the application of the model are given.

The equations presented in the sequel are often given in the form valid for a network. In a network any section connects two nodes, one of which is always denoted as 1 and the other as 2; it is determined by the user which node is taken as node 1 and which as node 2. Discharges and loads are taken to be positive if transferring mass from node 1 to node 2.

The numerical method is based on the use of both the mass conservation equation and the equation of motion in the section, and the use of the conservation equation (stating that the sum of the discharges is 0) in the nodes.

2.2 Flow

2.2.1 The unsteady flow equations

The DUFLOW package is based on the one-dimensional partial differential equation that describes non-stationary flow in open channels (Abbott, 1979; Dronkers, 1964).

These equations, which are the mathematical translation of the laws of conservation of mass and of momentum read:

$$B\frac{\partial H}{\partial t}+\frac{\partial Q}{\partial x}=0 \tag{1}$$

and

$$\frac{\partial Q}{\partial t}+gA\frac{\partial H}{\partial x}+\frac{\partial(\alpha Qv)}{\partial x}+\frac{g|Q|Q}{C^2AR}=b\gamma w^2\cos(\Phi-\phi) \tag{2}$$

while the relation:

$$Q=v\cdot A$$

holds and where:

t	time [s]
x	distance as measured along the channel axis [m]
$H(x, t)$	water level with respect to reference level [m]
$v(x, t)$	mean velocity (averaged over the cross-sectional area) [m/s]
$Q(x, t)$	discharge at location x and at time t [m^3/s]

$R(x, H)$	hydraulic radius of cross-section [m]
$A(x, H)$	cross-sectional flow area [m^2]
$b(x, H)$	cross-sectional flow width [m]
$B(x, H)$	cross-sectional storage width [m]
g	acceleration due to gravity [m/s^2]
$C(x, H)$	coefficient of De Chézy [m^2/s]
$w(t)$	wind velocity [m/s]
$\Phi(t)$	wind direction in degrees [degrees]
$\phi(x)$	direction of channel axis in degrees, measured clockwise from the north [degrees]
$\gamma(x)$	wind conversion coefficient [-]
α	correction factor for non-uniformity of the velocity distribution in the advection term, defined as: $\alpha = \frac{A}{Q^2}\int v(y,z)^2\,dydz$ where the integral is taken over the cross-section A. [m^2]

This mass equation (1) states that if the water level changes at some location this will be the net result of local inflow minus outflow. The momentum equation (2) expresses that the net change of momentum is the result of interior and exterior forces like friction, wind and gravity.

For the derivation of these equations it has been assumed that the fluid is well-mixed and hence the density may be considered to be constant.

The advection term in the momentum equation:

$$\frac{\partial(\alpha Q v)}{\partial x} \tag{4a}$$

can be broken into

$$\alpha\left(2\frac{Q}{A}\frac{\partial Q}{\partial x} - \frac{Q^2}{A^2}\frac{\partial A}{\partial x}\right) \tag{4b}$$

The first term represents the impact of the change in discharge. The second term which expresses the effect of change in cross-sectional flow area, is called the Froude term. In case of abrupt changes in cross-section this Froude term may lead to computational instabilities.

In chapter 3 is described how to manipulate this term.

2.2.2 Discretization of the unsteady flow equations

Equations (1) and (2) are discretized in space and time using the four-point implicit Preissmann scheme.

Defining a section Δx_i from node x_i to node x_{i+1} and a time interval Δt from time $t = t^n$ to time $t = t^{n+1}$, then discretization of the water level H can be expressed as:

$$H_i^{n+\theta} = (1-\theta)H_i^n + \theta H_i^{n+1} \tag{5}$$

at node x_i and time $t + \theta\Delta t$

and

$$H_{i+1/2}^n = \frac{H_{i+1}^n + H_i^n}{2} \tag{6}$$

in between nodes x_i and x_{i+1} at time t.

In a similar way other dependent variables can be approached.

The transformed partial differential equations can be written as a system of algebraic equations by replacing the derivatives by finite difference expressions. These expressions approximate the derivatives at the point of references ($x_{i+1/2}$, $t^{n+\theta}$) as shown in figure 2.1.

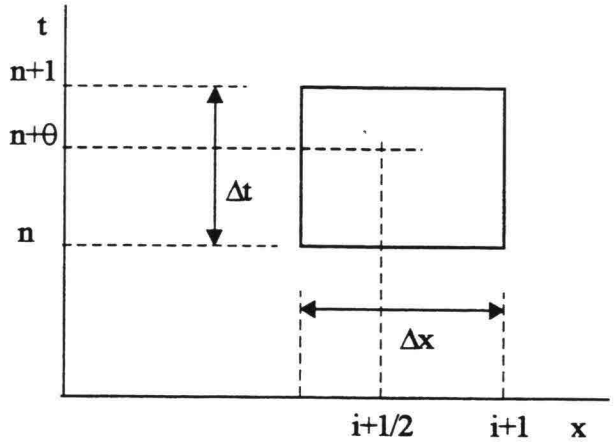


Figure 2.1 The Four Point Preismann scheme

Equation (1) is transformed into:

$$B^*_{i+1/2} \cdot \frac{H^{n+1}_{i+1/2} - H^n_{i+1/2}}{\Delta t} + \frac{Q^{n+\theta}_{i+1} - Q^{n+\theta}_i}{\Delta x_i} = 0 \qquad (7)$$

and equation (2) into:

$$\frac{Q^{n+1}_{i+1/2} - Q^n_{i+1/2}}{\Delta t} + \frac{gA^*_{i+1/2}\left(H^{n+\theta}_{i+1} - H^{n+\theta}_i\right)}{\Delta x_i} + \frac{\alpha\left(\frac{Q^n_{i+1}}{A^*_{i+1}} Q^{n+1}_{i+1} - \frac{Q^n_i}{A^*_i} Q^{n+1}_i\right)}{\Delta x_i} + \frac{g\left(Q^{n+1}_{i+1/2}\left|Q^n_{i+1/2}\right|\right)}{\left(C^2 AR\right)^*_{i+1/2}} = b^n \gamma(w^{n+1}_{i+1/2}) \cos(\Phi^{n+1} - \phi) \qquad (8)$$

The * (like in $B^*_{i+1/2}$) expresses that these values are approximated at time $t^{n+\theta}$. This discretization is of second order in time and place if the value $\theta = 0.5$ and it can be shown that in this case the discretized system is mass-conservative. In most applications, a somewhat larger θ-value, such as 0.55 is used in order to obtain a better stability (Roache, 1972).

The values indicated with (*) are computed using an iterative process.

For example, a first approximation of B is

$$B^* = B^n$$

which is adjusted in subsequent iteration steps:

$$B^* = \frac{\left(B^n + B^{n+1,*}\right)}{2}$$

where $B^{n+1,*}$ is the new computed value of B^{n+1}.

So finally, for all channel sections in the network two equations are formed which have Q and H as unknowns on the new time level t^{n+1}:

$$Q_i^{n+1} = N_{11}H_i^{n+1} + N_{12}H_{i+1}^{n+1} + N_{13} \tag{9a}$$

$$Q_{i+1}^{n+1} = N_{21}H_i^{n+1} + N_{22}H_{i+1}^{n+1} + N_{23} \tag{9b}$$

2.2.3 Boundary and initial conditions

For a unique solution of the set of equations additional conditions have to be specified at the physical boundaries of the network and at the sections defined as hydraulic structures. Structures are discussed in paragraph 2.2.4.

The user-defined conditions at the physical boundaries may be specified as levels, discharges or a relation between both. For instance a (tidal) elevation H, a discharge Q, or a so-called Q-H relation. The selection of boundary conditions is discussed in more detail in paragraph 2.5.

At internal junctions the (implicit) condition states that the water level is continuous over such a junction node, and that the flows towards the junction are in balance since continuity requires:

$$\sum_{J=1}^{JJ} Q_{j,i} + q_i = 0 \tag{10}$$

where:

i	indication for the junction node
$Q_{j,i}$	discharge from node j to node i
q_i	additional or lateral flow to node i

The above equations are solved at each time step. They are transformed into a system of (linear) equations for the water levels by substitution of the equations (9a) and (9b). After the water levels are computed using a standard solution method for systems of linear equations, the discharges are found by substituting the computed water levels into equations (9a) and (9b).

Equation (10) is not used in nodes where a water level is prescribed as boundary condition. In such a node no equation is needed because the water level is already known. Discharge boundary conditions are taken into account as the additional flow q_i.

To start the computations, initial values for H and Q are required.

These initial values must be provided by the user; they may be historical measurements, obtained from former computations or just a first reasonable guess.

Additionally wind stress and rainfall conditions can be specified.

2.2.4 Structures

2.2.4.1 General about structures

Various types of control structures can be defined such as weirs, culverts, siphons and pumping stations, which cover most of the control structures existing in real-life systems.

At weirs and other structures discharges and levels can be controlled by manipulating the gates. DUFLOW allows for specification of such an operation using the so-called trigger conditions: depending on flow conditions at specified locations in the network, parameters such as the width of the weir, the level of the sill, etc. can be adjusted during the computation. These conditions are treated in more detail in chapter 3.

A common characteristic of structures is that the storage of water inside the structure is negligible compared with the storage in the open channels.
The definition of flow direction in a structure is the same as the definition in ordinary channel sections, flow from the begin node to the end node is assumed to be possitive.

Any structure is defined between two nodes i and j and the discharge in the structure is denoted simply as Q.

2.2.4.2 Weirs

The discharge over a weir depends on the water level at both sides, the level of the sill, type of structures and the flow condition (free surface or submerged flow).

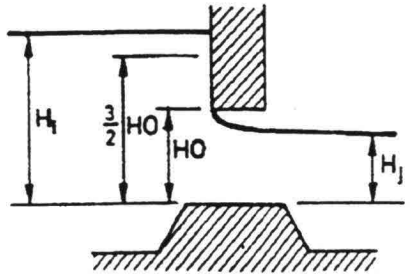
Figure 2.2 shows the seven hydraulic conditions of flow over a weir that can be distinguished, where it is assumed that $H_i > H_j$. If $H_i < H_j$, the picture is symmetrical with figure 2.2 except for the loss coefficient, which need not be symmetrical.

The general equation for the discharge over the weir is
where:

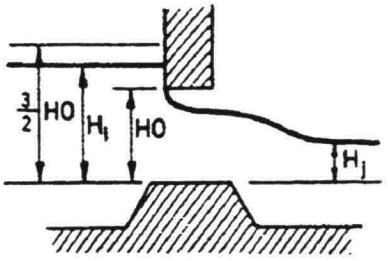
$$Q^{n+1} = \mu B H \sqrt{2g\Delta H}$$

B	width of the weir,
μ	the discharge reduction or loss coefficient,
H	depth over the sill,
ΔH	difference in head.

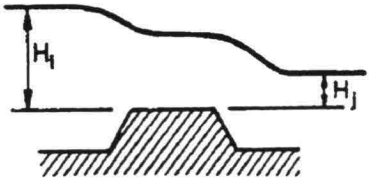
These quantities, which may be different depending on the actual type and direction of flow, are given in table 2.1.



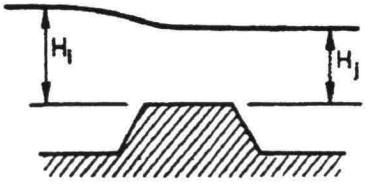
I : $H_1 > \frac{3}{2} HO$
$H_j < HO$



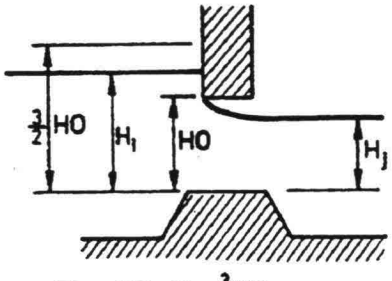
II : $HO < H_1 < \frac{3}{2} HO$
$H_j < \frac{2}{3} H_1$



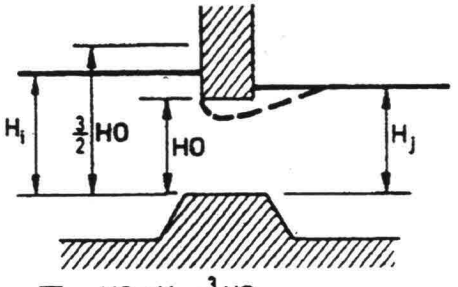
III : $H_1 < HO$
$H_j < \frac{2}{3} H_1$



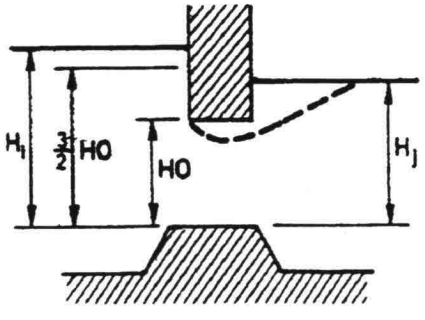
IV : $H_1 < HO$
$H_j > \frac{2}{3} H_1$



V : $HO < H_1 < \frac{3}{2} HO$
$\frac{2}{3} H_1 < H_j < HO$



VI : $HO < H_1 < \frac{3}{2} HO$
$H_j > HO$



VII : $H_1 > \frac{3}{2} HO$
$H_j > HO$

Figure 2.2 Weir Flow conditions, covered in DUFLOW

flow condition	μ	H	ΔH
I	μ_0	HO	$H_i^{n+1} - HO$
II	μ_t	$\frac{2}{3}H_i^{n+1}$	$\frac{1}{3}H_i^n$
III	μ_v	$\frac{2}{3}H_i^{n+1}$	$\frac{1}{3}H_i^n$
IV	μ_v	H_i^n	$H_i^{n+1} - H_j^{n+1}$
V	μ_t	H_i^n	$H_i^{n+1} - H_j^{n+1}$
VI	μ_t	HO	$H_i^{n+1} - H_j^{n+1}$
VII	μ_0	HO	$H_i^{n+1} - H_j^{n+1}$

Table 2.1 Coefficients in equation (11)

The parameters are defined as:

H_i, H_j water depth over the sill, respectively at the beginning and at the end of the section

HO height of gate opening

μ_0 loss coefficient, gate flow

μ_v loss coefficient, free surface flow

μ_t loss coefficient, transition between μ_0 and μ_v, i.e.

$$\mu_t = \mu_v + 2\left(\frac{H_i}{HO} - 1\right)(\mu_0 - \mu_v) \tag{12}$$

Note that these formulas are approximations of the real physical situation and can be calibrated using loss-coefficients μ_0 and μ_v.

Equation (11) is linearized and brought into the form generally used for structures:

$$Q^{n+1} = N_1 H_i^{n+1} + N_2 H_j^{n+1} + N_3 \tag{13}$$

2.2.4.3 Culverts

Culverts are governed by the same flow equations as weirs, only with a friction term added. This resistance is thought to be coupled serially with the weir, at the upstream side.

The equation for the resistance is based on the De Chézy formula:

$$Q = AC\sqrt{R\frac{\Delta H}{L}} \tag{14}$$

where ΔH is the head loss due to friction, and L is the length of the culvert as prescribed by the user. The quantities A, C, and R are defined in paragraph 2.2.

The resistance formula is linearized as:

$$|Q^n|Q^{n+1} = \frac{A^2C^2R\Delta H}{L}$$

or

$$Q^{n+1} = \frac{A^2 C^2 R}{L|Q^n|} \Delta H = F \Delta H$$

Serial coupling means that a dummy node I is inserted between the nodes i and j. The formula for the discharge between the begin node i and the dummy node I is based on the resistance formula:

$$Q^{n+1} = F.\left(H_i^{n+1} - H_I^{n+1}\right).$$

and the formula for the discharge between node I and the end node j is equal to the weir formula:

$$Q^{n+1} = N_1 H_I^{n+1} + N_2 H_j^{n+1} + N_3$$

H_I is eliminated and the following result is obtained:

$$Q^{n+1} = \frac{FN_1}{F+N_1} H_i^{n+1} + \frac{FN_2}{F+N_1} H_j^{n+1} + \frac{FN_3}{F+N_1}$$

The general expression for the culvert is:

$$Q^{n+1} = N_1^* H_I^{n+1} + N_2^* H_j^{n+1} + N_3^* \qquad (15)$$

where the coefficients for flow in both directions are:

$$N_1^* = \frac{F}{F + \max(N_1, -N_2)} N_1$$

$$N_2^* = \frac{F}{F + \max(N_1, -N_2)} N_2$$

$$N_3^* = \frac{F}{F + \max(N_1, -N_2)} N_3$$

2.2.4.4 Siphons

A siphon is a fully filled closed conduit to transport water over obstacles (hills, structures). In case of flow the equation for Q is the same as case VII in paragraph 2.4.2 combined with resistance in the same way as in paragraph 2.4.3.

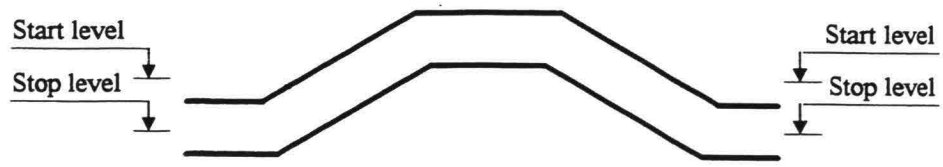


Figure 2.3 Operation levels of a siphon

A siphon is in full operation when the upstream and downstream water levels are above user defined start values. The siphon runs dry when at one of both sides the water level is below an user defined switch-off or stop level.

2.2.4.5 Pumps

Like a siphon a pump is assumed to be in full operation or not at all.

In the flow state the discharge is simply a constant Q_p prescribed by the user. It is assumed that Q_p is positive, i.e. carrying water from begin node i to end node j.

Operation is controlled by the water level at upstream or begin node i. If this water level drops below an user defined switch-off level (stop level) then the discharge is set to zero; if it rises above the switch-on level (start level) given by the user, the discharge is set to pump capacity Q_p (see figure 2.4).

Note that the switch-on level must be higher than the switch-off level. For reasons of stability it may be necessary to define a reasonable additional storage capacity by introducing an extra section at node i.

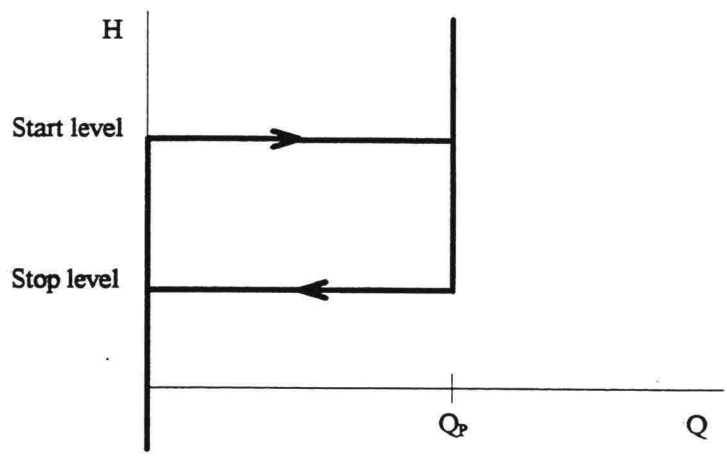


Figure 2.4 Dependence of pump operation on upstream water level

2.3 Quality

2.3.1 The mass transport equation

The quality part of the DUFLOW package is based upon the one dimensional transport equation. This partial differential equation describes the concentration of a constituent in an one dimensional system as function of time and place.

$$\frac{\partial(BC)}{\partial t} = -\frac{\partial(QC)}{\partial x} + \frac{\partial}{\partial x}\left(AD\frac{\partial C}{\partial x}\right) + P \qquad (16)$$

where:

C	constituent concentration [g/m^3]
Q	flow [m^3/s]
A	cross-sectional flow area [m^2]
D	dispersion coefficient [m^2 /s]
B	cross-sectional area [m^2]
x	x-co-ordinate [m]
t	time [s]
P	production of the constituent per unit length of the section [g/m.s]

The production term of the equation includes all physical, chemical and biological processes to which a specific constituent is subject to. In paragraph 2.3.4 a detailed description of the way the processes are dealt with is presented. For the time being in this paragraph the overall production rate for constituent C is represented using the lumped variable P.

Equation (16) has to be solved numerically. The solution technique selected is the method used in model flows (Booij, 1978). In order to apply this method equation (16) is rewritten as:

$$\frac{\partial S}{\partial x} + \frac{\partial(BC)}{\partial t} - P = 0 \qquad (17)$$

in which S is the transport (quantity of the constituent passing a cross-section per unit of time):

$$S = QC - AD\frac{\partial C}{\partial x} \qquad (18)$$

Equation (18) describes the transport by advection and dispersion. Equation (17) is the mathematical formulation of the mass conservation law, which states that the accumulation at a certain location x is equal to the net production rate minus the transport gradient.

In this form the transport equation closely resembles the equations for the flow so that similar numerical approximations may be applied.

2.3.2 Discretization of the mass transport equation

The numerical method used to solve the transport equations (17) and (18) was adopted from the model flows (Booij, 1978). The method is unconditionally stable and shows little numerical dispersion. Furthermore the method perfectly fits to the discretization of the flow equations. Similar to the flow computations, adjacent sections may be different in length.

In the flow computation the discharges at each end of a section are expressed as (linear) functions of the water levels at both ends. Here we express the transports at both ends (S_1 and S_2) into the concentrations at both ends (c_1 and c_2). To obtain such expressions we apply Galerkin's method.

The mass conservation equation (17) is integrated over the section, multiplied with a weighting function ψ_1 or ψ_2:

$$\int_0^{\Delta x} \psi_i \left[\frac{\partial S_i}{\partial t} + \frac{\partial (B_i C)}{\partial x} - P_i \right] dx = 0 \tag{19}$$

results into:

$$\psi_i S_i \Big|_0^{\Delta x} + \int_0^{\Delta x} \left[\psi_i \frac{\partial (B_i C)}{\partial t} - \frac{\partial \psi_i}{\partial x} \left(QC - A_i D \frac{\partial C}{\partial x} \right) - \psi_i P_i \right] dx = 0 \tag{20}$$

In each section we distinguish two weighting functions:

$$\psi_1 = 1 - \frac{x}{\Delta x} \tag{21}$$

$$\psi_2 = \frac{x}{\Delta x} \tag{22}$$

C is assumed to vary linearly within the section Δx, i.e.:

$$C = \psi_1 c_1 + \psi_2 c_2 \tag{23}$$

in which c_1 and c_2 are the concentrations at the beginning and the end of the section.

Solution of equation (20) for i = 1,2 results into:

$$-S_1 + \frac{\Delta x}{3} \frac{\partial B_1 c_1}{\partial t} + \frac{\Delta x}{6} \frac{\partial B_2 c_2}{\partial t} + \frac{Q_1 c_1 + Q_2 c_2}{2} - A_1 D \frac{c_2 - c_1}{\Delta x} + - \frac{\Delta x}{3} P_1 - \frac{\Delta x}{6} P_2 = 0 \tag{24}$$

$$+S_2 + \frac{\Delta x}{6} \frac{\partial B_1 c_1}{\partial t} + \frac{\Delta x}{3} \frac{\partial B_2 c_2}{\partial t} + \frac{Q_1 c_1 + Q_2 c_2}{2} + A_2 D \frac{c_2 - c_1}{\Delta x} + - \frac{\Delta x}{6} P_1 - \frac{\Delta x}{3} P_2 = 0 \tag{25}$$

In which indices 1 and 2 refer to the beginning and the end of the section.

In equations (24) and (25) discretization with respect to x has taken place: the quantities S and C are still continuous functions of t.

Discretization of the equations (24) and (25) with respect to time using a time step Δt results into:

$$S_1^+ = \frac{(1-\theta)}{\theta} S_1^- + \frac{\Delta x}{3\theta}\left(\frac{B_1^+ c_1^+ - B_1^- c_1^-}{\Delta t}\right) + \frac{\Delta x}{6\theta}\left(\frac{B_2^+ c_2^+ - B_2^- c_2^-}{\Delta t}\right) + \\ + \frac{\theta Q_1^+ c_1^+ + \theta Q_2^+ c_2^+ + (1-\theta) Q_1^- c_1^- + (1-\theta) Q_2^- c_2^-}{2\theta} + \\ - A_1 D \frac{\theta c_2^+ - \theta c_1^+ + (1-\theta) c_2^- - (1-\theta) c_1^-}{\Delta x \theta} - \frac{\Delta x}{3\theta} P_1 - \frac{\Delta x}{6\theta} P_2 \qquad (26)$$

$$S_2^+ = \frac{(1-\theta)}{\theta} S_2^- + \frac{\Delta x}{6\theta}\left(\frac{B_1^+ c_1^+ - B_1^- c_1^-}{\Delta t}\right) + \frac{\Delta x}{3\theta}\left(\frac{B_2^+ c_2^+ - B_2^- c_2^-}{\Delta t}\right) + \\ + \frac{\theta Q_1^+ c_1^+ + \theta Q_2^+ c_2^+ + (1-\theta) Q_1^- c_1^- + (1-\theta) Q_2^- c_2^-}{2\theta} + \\ - A_2 D \frac{\theta c_2^+ - \theta c_1^+ + (1-\theta) c_2^- - (1-\theta) c_1^-}{\Delta x \theta} - \frac{\Delta x}{6\theta} P_1 - \frac{\Delta x}{3\theta} P_2 \qquad (27)$$

The indices + and - refer to the present and last time step respectively. The weighting factor with respect to time is θ. Using a value of θ = 1 results into a fully implicit method. In paragraph 2.5 some practical considerations on the selection of θ are given.

Unknowns in the above equations are: c_1^+, c_2^+, S_1^+ and S_2^+. Normally but not always (see section 2.3.3.4 on "decoupling"), c_1^+ and c_2^+ are equal to the concentrations in the adjacent nodes so that S_1^+ and S_2^+ can be expressed into the concentrations at the nodes just like the discharges are expressed in the water levels at the nodes by (9a) and (9b):

$$S_i^{n+1} = N_{11} c_i^{n+1} + N_{12} c_{i+1}^{n+1} + N_{13}$$

$$S_{i+1}^{n+1} = N_{21} c_i^{n+1} + N_{22} c_{i+1}^{n+1} + N_{23}$$

Using these equations together with a mass balance over the nodes results into a set of linear equations which can be solved, see paragraph 2.4. The figure below shows at which locations in the system matrix of the set of equations for C the submatrix N given above fits in. The coefficients N_{11}, N_{12}, N_{21} and N_{22} are found in the matrix E on the left hand side, the coefficients N_{13} and N_{23} appear in the right hand side (F):

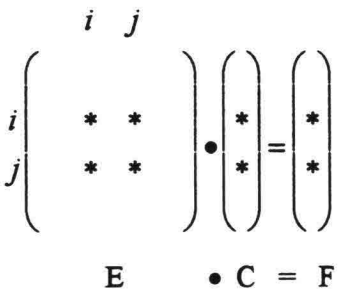


Figure 2.5 Structure of the set of equations

2.3.3 Initial and boundary conditions

2.3.3.1 Initial conditions

To start the computations, initial values for all state variables (concentrations) are required. These initial concentrations must be supplied for each node. They can be based on historical measurements, obtained from former computations or from a first reasonable guess.

In the last case the user should realise that the first part of the simulation period may result into non realistic results. The time during which the effect of the initial conditions is perceptible is controlled by the transport rates and the characteristic time constants for the reactions. As a rule of thumb for a single section, if only advection is taken into account, the impact of the initial conditions is completely gone after three times the residence time in this particular section.

As the exchange rates between the sediment layer and the overlying water are rather low especially for the concentrations in the sediment reasonable initial values have to be used. If no reasonable initial values are available it is recommended to perform some initial simulations from which the results, at the end of the simulation period can be used as initial values for a next run.

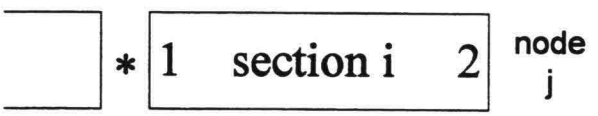
2.3.3.2 Boundary conditions

The way boundary conditions are treated in the quality part of DUFLOW is more complicated than for the flow part. Therefore this subject is discussed here with some more detail. In the first place we have to distinguish between two types:

- the physical boundaries of the system.
- the internal nodes in the system.

At the begin or end nodes in the network, in the flow part, a water level or Q boundary can be applied. In the quality part at these locations a concentration boundary can be used. The different combinations of C and Q-H -boundaries are shown in the next paragraphs.

2.3.3.3 Physical boundaries of the network



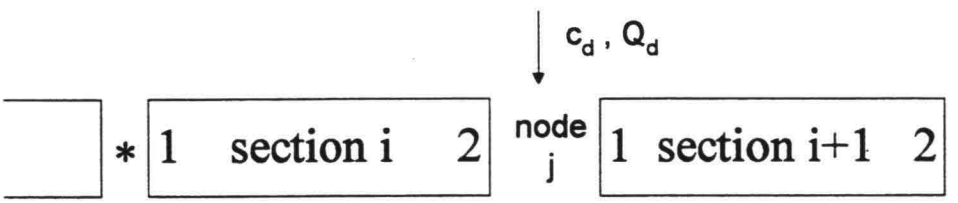
- *H-boundary combined with C-boundary*
  If the flow direction at node j is towards the environment: $S_2 = Q_2 c_2$
  If the flow direction at node j is into the network for S_2 equation (27) is used and in the mass balance equation for node j an additional load equal to $Q_2 c_b$ is added. Where c_b is the boundary concentration.
- *Q-boundary combined with C-boundary*
  If the flow direction at node j is towards the environment: $S_2 = Q_2 c_2$

If the flow direction at node j is into the network for S_2 equation (27) is used and in the mass balance equation for node j an additional load equal to $Q_b\ c_b$ is added. Where Q_b and c_b are the flow and concentration at the boundary respectively.

- *Fixed concentration boundaries*
  If a C-boundary is defined at node j, without using a H or Q boundary. The specified concentration c_b is substituted in equation (27).
- *Dead end sections of the network*
  If no H, Q or C boundaries conditions are specified at node j; $S_2 = 0$.
  Physically this situation represents a dead end section of the network.

  In case of physical H-boundaries, Q-boundaries and combined Q-H boundaries, a concentration boundary for every defined dissolved substance is compulsory.

2.3.3.4 Internal nodes



- *Q-boundary combined with C-boundary*
  This situation represents a discharge with a flow rate Q_d and a concentration c_d at node j in the network. If Q_d is positive (flow into the network): in the mass balance equation for node j an additional load $Q_d\ c_d$ is added.

  If Q_d is negative an additional load equal - $Q_d c_{node}$ is added to the mass balance equation for node j. Where:
  $c_{node} = c_{1,i+1}$ if the flow in the network is positive (from i to i+1)
  $c_{node} = c_{2,i}$ if the flow direction is from i+1 to i

  For the calculations of the mass transport in section i and i+1, two options are available in this case.

  - If the calculation is performed using the option decouple (see calculation definition, paragraph 3.4.1.1, Decouple = Yes).
    $S_{2,i} = Q_{2,i} c_{2,i}$ if flow is from i to i+1
    $S_{1,i+1} = Q_{1,i+1} \cdot c_{1,i+1}$ if flow is from i+1 to i
    This means that the dispersion coefficient in the section downstream the discharge is set equal to zero (0).
  - Default this decouple option is not used and the mass transport is calculated using equations (26) and (27).

This decouple option can be used to prevent flattening of steep concentration gradients at nodes where a discharge is located.

In case of a Q-boundary at an internal node, the user should either define a concentration boundary for every defined dissolved substance or define no concentration boundary at all for that internal node:

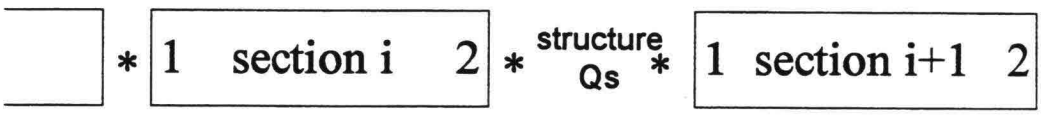
- Q-boundary combined with c-boundary for every dissolved substance.
  Water flowing in- and out of the model as defined above
- Q-boundary, no c-boundary.
  Water flowing in- and out of the model contains no dissolved matter, c=0. This feature may cause confusion but is added to enable the user to model evaporation.

2.3.3.5 Loads

At all nodes in the network an additional load can be applied, which means that this load is added to the mass balance equation of the node. A load is expressed as a mass unit per second. The user should pay attention to the use of proper dimensions.

2.3.3.6 Structures

In structures no processes take place. Mass is only passed between the connected nodes.



If flow is from i to i+1: $S_1 = S_2 = Q_s\, c_{2,i}$

If flow is from i+1 to i: $S_1 = S_2 = Q_s\, c_{1,i+1}$.

2.3.4 Processes

2.3.4.1 General

In DUFLOW the mathematical formulations describing the processes can be supplied by the user. They have to be supplied in the model file (*.mod), which can be created or edited using the user interface (see paragraph 3.8). For this purpose a special process description language has been developed. The set of process descriptions supplied by the user can be compiled using DUPROL. This compiler converts the descriptions into data, which are read by the calculation part of DUFLOW. In this way the user can change the process descriptions or define a new set of descriptions without changing and recompiling the source code of the calculation part of DUFLOW. A full explanation of the syntax of the language and the use of DUPROL is given paragraph 3.8.

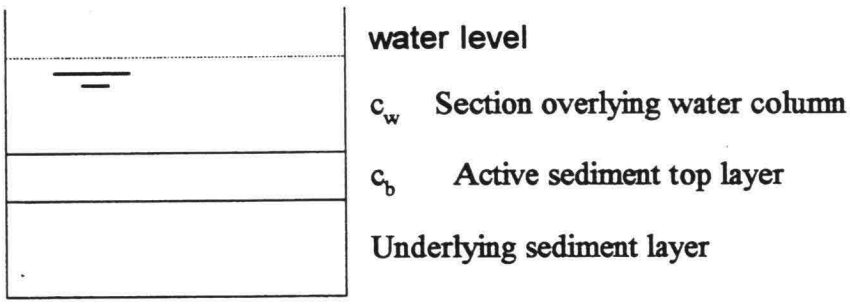
In the computational part the process descriptions are combined with the transport equations. Actually the lumped variable P introduced in equation (16) is filled in. In the preceding paragraph it was shown how the numerical solution of the mass transport equation resulted into a matrix equations. In this equation P is in the right hand side of the equation, which means that all process descriptions are evaluated using the values of the state variables at the previous time step.

This explicit evaluation of the process description does have some consequences for the time step to be used. This subject is dealt with in paragraph 2.6.

If more then one variable is defined in the process description file, for each variable the matrix equation is solved. This matrix equation is solved using LUD decomposition. Using this method the left hand side matrix has to be manipulated only once.

2.3.4.2 Bottom variables

In the process description file two types of state variables can be declared. Water type state variables have been dealt with in the previous part of this paragraph. For bottom state variables the transport part of equation (16) is not calculated. In this way for each section a bottom state variable can be defined to describe the interaction between the sediment and the overlying water column. The depth of the bottom layer and the transport across the water/sediment interface must be supplied by the user in the process description file.



In this example we consider an active sediment top layer. The thickness of this top layer is considered to be constant in time. If also the porosity of the sediment is assumed to be constant the exchange flux of a dissolved constituent between the sediment interstitial water and the water column can be expressed as:

$$F_{ex} = \frac{E_{ex}}{H_b}\left(c_w - \frac{c_b}{por}\right)$$

where:

F_{ex}	the exchange flux between water and sediment [g/ m^2. day]
E_{ex}	diffusive exchange rate [m^2 /day]
c_w	the constituent concentration in the overlying water column [g/ m^3]
c_b	the constituent concentration in the sediment [g/m^3]
H_b	thickness of the sediment top layer [m]
por	the porosity of the sediment top layer [-]

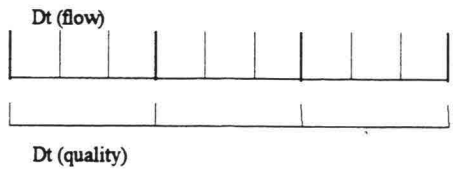
The description of the exchange flux has to be added to the differential equation describing the processes of the constituent. Both the depth of the sediment top layer and the exchange rate should be declared as a parameter.

In principle more complex descriptions also can be used to describe the sediment water interactions. Another example is given in eutrof2 (see appendix D).

If necessary the sediment can be subdivided into more layers. The exchange between the distinguished layers can be expressed in the same way.

2.3.5 Options for computation

In DUFLOW four options for computation are available, see paragraph 3.5. Flow and quality can be calculated simultaneously or separately. In the last case the flow conditions (information like flow, flow width, etc.) are stored in the *.dmp file and read during simulation of quality. Different time steps can be used for calculation of flow and quality. The time step used for quality must be a multiple of the time step used for flow.



For calculation of quality the flow is averaged over N_{flow} (flow) time steps, where N_{flow} is the number of flow time steps in one quality time step.

Using the option Box only the processes are simulated. Actually the transport related terms in equation (16) are eliminated. The processes are simulated in all sections of the network. Hence this option enables to evaluate the relative contribution of transport compared to the processes in the entire network. Furthermore this option can be usefully for testing during model development.

2.4 Solving the set of equations

The network as a whole is a system of sections and nodes, where each channel section and control structure is considered as a separate item. Each branch and node of the network has a unique identification number, assigned by the user. The structure of the system is implicitly defined by the user specification of node numbers at both ends of each section.

The number of unknowns is in principle equal to 2*J+I, where J is the number of sections and I is the number of nodes; in each branch the unknowns are the flows at both ends and at each node the water level. At structures the flow at begin and end node is the same.

The number of equations is also 2*J+I; for each channel section j two equations are derived, following from the mass and momentum equation (9). At structures only the momentum equation is applied (11, 14) as the mass equation can be neglected because of the no-storage condition. At each node there is a balance equation for the flows (10), since it is assumed in accordance with the four-point method that the storage of water takes place inside the branches, and not at the nodes. At boundary defined nodes an additional condition can be specified; thus there is one equation for each node.

Since the numerical scheme is implicit, a set of linear equations has to be solved for each time step.

Q_i and Q_{i+1} represent the discharge through cross-sections at the begin and end node of each section, and H_i and H_{i+1} represent the water level at these nodes.

After substitution of equation (9, 11, 14) into (10) a system of linear equations results in which the water levels are the unknown variables:

$$\sum_j M_{ij} H_j = R_i \qquad (30)$$

This system of equations is built up in the program section by section. For the channel section with nodes i and j, the coefficients N_{11}, N_{12}, N_{21} and N_{22} in equation (9) contribute to the matrix coefficients M_{ii}, M_{ij}, M_{ji} and M_{jj} respectively, and the coefficients N_{13} and N_{23} to R_i and R_j respectively. For control structures similar conditions hold.

In this way the structure of the network is recognisable in the structure of the system of equations, see figure 2.6:

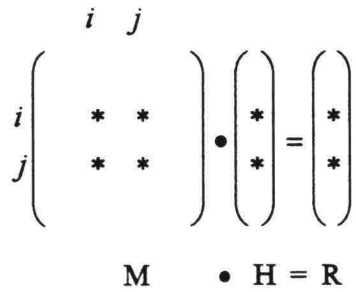


Figure 2.6 Structure of the set of equations

In most cases the matrix N is sparse, which means that most elements are zero with non-zero elements scattered over the entire matrix.

The solution method is based on LUD decomposition. For details of this method the user is referred to numerical handbooks. For instance (Numerical Recipes) a comprehensive description of lud decomposition is given.

2.5 Practical considerations

In this paragraph a few suggestions are given concerning practical modelling aspects using the DUFLOW package. These suggestions are valid for any open channel network and are described in even more detail in the literature. Here only a brief outline is given.

Defining a model for a specific situation requires decisions like:

- extent of the computational region, in space and time;
- nature of boundary conditions;
- schematisation of channel sections, structures etc.;
- space and time discretization;
- type of quality model selected and level of detail used in process descriptions.

Extent of the computational region, in space and time

DUFLOW is a predictive model, especially suited for the simulation of changes in existing systems. Simulation results of the actual situation can be verified, which is not the case for the simulation of the new situation.

Extrapolation of simulation results for quality to new situations also should be done carefully. A model might give an outstanding description of the present situation after calibration. But due to changing circumstances in the new situation the relative importance of certain processes may change. For example after reduction of the external phosphorus load the release of phosphorus from the sediment may become important.

In particular the boundaries must be chosen with great care in cases where a change in the system may effect a boundary condition which in turn may influence the hydraulic conditions and quality in the region of interest. Since the same boundary conditions are applied in the present and new situations, this may lead to erroneous results in the simulation of future changes. So one should take care that:

- either any change in the system does not affect a boundary condition;
- or the boundary condition does not influence the state in the region of interest.

For example if a dam is planned in a river basin and one wants to predict the change in flood level at a location downstream of this dam, the upstream boundary should be chosen so far upstream that at that place no influence of the dam is expected.

The downstream boundary must be situated so far downstream that a wave reflected at this boundary is damped out at the location of interest. These choices can be verified by variations in the model: in case of the upstream boundary a computation with and without dam can be compared; in the case of the downstream boundary two computations with different locations of this boundary can be compared.

Nature of boundary conditions

When the location of a boundary is determined, the next choice is what type of boundary condition (water level, discharge of H-Q- relation) is to be used. The best choice is that quantity or relation that is least sensitive to the state in the model itself.

Thus the upstream boundary condition in a river is preferably a discharge whereas the downstream boundary condition should be a water level if the river flows into a lake or sea, or a H-Q relation based on uniform flow if the downstream boundary is somewhere along the river.

Note that a dead-end (Q = 0 permanently) is the default boundary condition in DUFLOW; such a boundary condition does not have to be mentioned explicitly in the program input.

When simulating quality a flow boundary in combination with a concentration boundary is used at the upstream boundaries of the network. At the downstream ends a concentration boundary condition is not necessary as long as the flow direction is directed towards the environment. In this case the upstream concentration is not influenced by the boundary condition. However if for some reason the direction of flow changes a concentration boundary should also be provided at these boundaries of the network.

Discharges at internal nodes in the network can be simulated using a Q and c boundary. However if the flow of the discharge is small compared to the flow in the receiving water also a load can be used.

Schematisation of channel sections, structures etc.

Very detailed schematisation of a network is seldom necessary due to the nature of the equations involved. Usually small changes in cross-sections have only little influence on the state in a region of interest.

It is useful to start with a rather crude model and to test the sensitivity of the model to small changes in cross-sections before going to a detailed description. This is also true for structures.

For instance it is not efficient to model every bridge or other obstacle as a separate structure. It is better to introduce an increased channel friction to compensate for this resistance. Only structures that considerably reduce the cross-section should be modelled explicitly.

In general the level of detail used in quality simulation depends on both the nature of the system and the type of issues subject to modelling. The time and spatial scales of the variations in the concentrations are important in this respect. These scales are controlled by the rates of the underlying processes and the variability of the discharges and the environmental conditions.

Of course also the desirable accuracy of the results plays an important role.

Note: at both ends of a structure a normal section must be defined, in other words, a structure can not be located at the boundary of the model.

Space and time discretization

For space and time steps similar arguments apply. Very detailed description is often unnecessary. The space step must be such that changes in cross-section are reasonably well followed; furthermore the space step must be a small fraction (say 1/40th, or less) of the wavelength, if any. Also the time step must be a small fraction of the wave period. A sensitivity analysis giving the influence of the size of the time step is always recommended and easily carried out.

The selection of space and time steps also influences the numerical dispersion introduced by discretization of the mass transport equation. This numerical dispersion leads to an additional smoothing of concentration gradients. The numerical dispersion for the solution method used can be approximated by:

$$E_{num} = \frac{u}{2}(1-2\theta)u\Delta t \tag{31}$$

For $\theta = 0.5$ this results into a numerical dispersion equal to 0. A value if $\theta < 0.5$ however may lead to a non positive solution, which may cause instabilities. A value of 0.55 is recommended.

As the production terms are calculated using the state at the preceding time step the selection of the time step used is also restricted by the process rates. As a rule of thumb Δt should be less than half of the characteristic time constant of the fastest process.

In the spatial schematisation first nodes are selected at the boundaries of the network, at junctions, and at both sides of a structure (immediately adjacent to the structure). Subdivisions of the branches are applied whenever a branch is longer than the desired space step; denser subdivisions are applied in a region with rapid changes in cross-section. Neighbouring river sections with different length are not disadvantageous with regard to accuracy since the Preissmann scheme is used.

Finally it must be stressed that applications by users without a proper understanding of phenomenon and/or without reliable verification increases the risk of erroneous results.

2.6 Limitations of the model

There are a number of inherent limitations in the equations and methods used in DUFLOW. The most important ones are summarised in this paragraph.

The equations are for one-dimensional flow. This means that water bodies with significantly different velocities in the vertical cannot be modelled. For instance the model is not suitable for stratified waters. Also the flow must be directed roughly parallel to the channel axis. Differences in flow velocity between the main channel and the flood plains can be taken into account by distinguishing the width of flow and the width of storage; however, if there are differences in water level between the main channel and the flood plains, it is better to model the two as separate (parallel) channels.

As mentioned, vertical density differences are not taken into account; also horizontal density differences are not modelled because the density is assumed to be constant throughout.

Although the equations underlying the model are valid in case of supercritical flow, the numerical solution method does not support supercritical flow, except inside some structures. Because subcritical flow is assumed there must be one boundary condition at each of the boundaries of the network.

The size of the application made by the user is restricted. Like in DUFLOW 1.00 a network up to 250 sections can be used. In order to estimate the memory needed for an application the program DUFDIM can be used. This program gives a rough estimation of the memory needed for the available computational options. Also an estimation of the maximum number of time steps to be calculated is provided, see appendix A. An extended version of the program without this restriction is available. For more information about this version please contact EDS.

3. Usage of the program

3.1 Introduction

This chapter is intended to be read by those who are using the DUFLOW package for the first time, but also is referred to appendix B (Getting started). More experienced users can run the program using only the information available on screen. Nevertheless the experienced users will find in this chapter a convenient reference book.

Paragraph 3.2 outlines the menu structure and the various conventions on how to use these menus are outlined. In paragraph 3.3 to 3.9 all the menu items and selections are discussed in detail.

The program is fully menu driven and user oriented, including flagging user errors and generating default values. Default values are also substituted when the actual value is meaningless or unlikely. Still no guarantee can be given that every error will be detected.

It is advised to save new data regularly to avoid loss of data if occasionally an error causes the program to stop. Control is always returned to the master menu.

The installation of the program on a computer is discussed in appendix A and is assumed here to be completed.

The package is started by entering

DUFLOW ↵

After display of the introduction screen the "master menu" which controls the program is shown.

3.2 Conventions and menu structure

3.2.1 Conventions for the use of the program

3.2.1.1 Conventions

- One character answers (like a choice in a menu) do not need a ↵ touch.
- Answering with only ↵ means:
  Question: default answer (first choice between brackets)
  In tables: accepting the displayed value
  In menus: choosing the high-lighted option
- Using tables the highlighted bar shows the actual value to edit.
  Use the cursor keys to move over the screen.
- Use ← (Backspace) and re-type to adjust values (displayed at the bottom of the screen).
- Toggling the = key recalls the last 10 given answers.

- Hit Esc to leave the present item of the program, so from a table to the choice bar at the bottom of the screen, from the choice bar to the menu, from the menu to the previous menu, up to the "master menu".

Conventions for the text in this chapter:

- "brackets" Text in double brackets refers to terms that are shown by DUFLOW on the screen.
- **Bold** Bold characters (except headings) refer to answers to be given by the user.

Definitions:

Levels	All levels are related to a user defined horizontal reference level. Positive values are above this reference level.
Depth	Distance between "Bottom level" and water level.
Section	A part of an open channel corresponding to an elementary step x in the numerical scheme (see chapter 2).
Structure	A flow control structure such as a weir, culvert, siphon, pump, etc.
Node	Junction between two or more sections or physical boundary of a section.

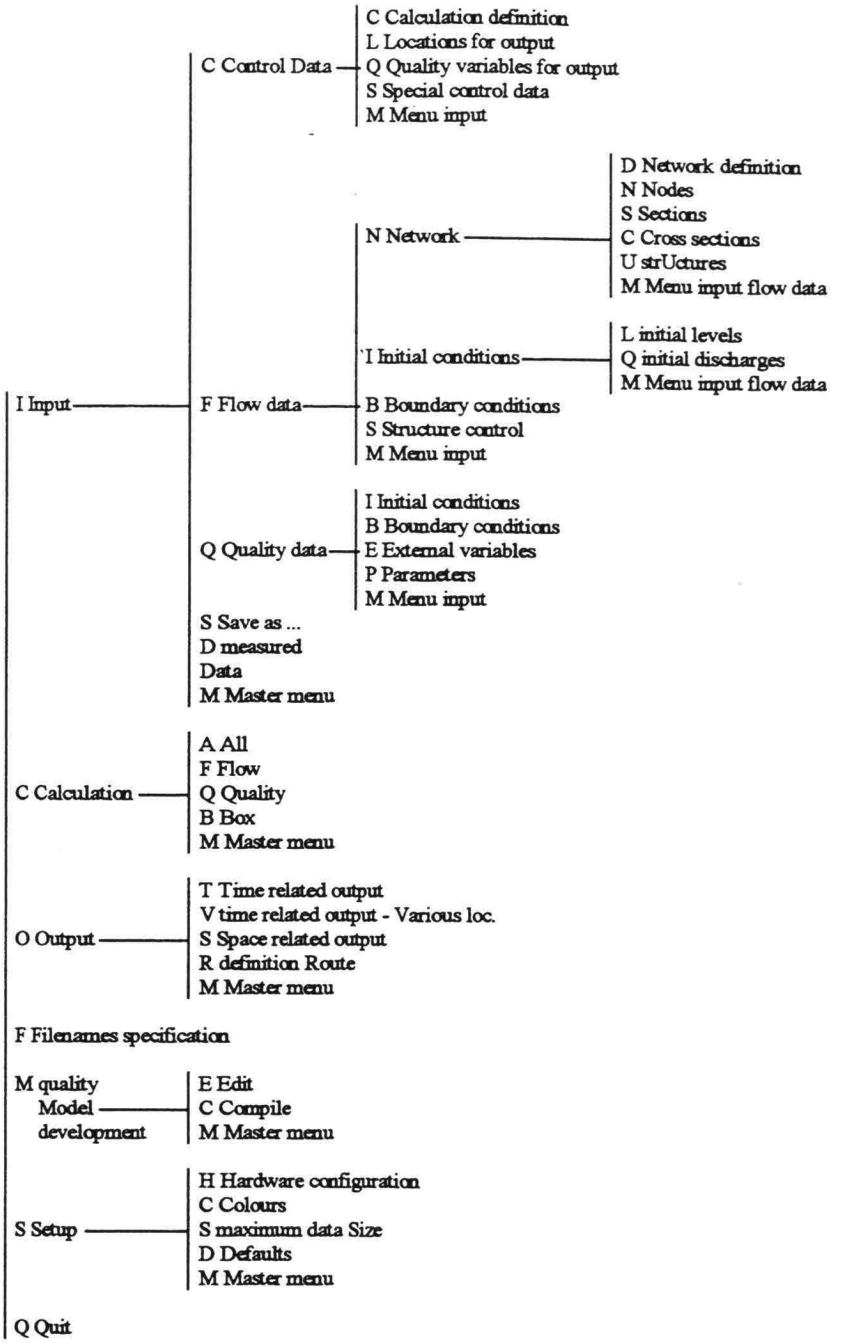
3.2.1.2 Menu structure

The menu structure of the DUFLOW package is shown on the next page.

3.2.1.3 Use of the menu

The interface towards the user consists off three parts:

- menus (see paragraph 3.2.2)
- checklists (see paragraph 3.2.3)
- tables (see paragraph 3.2.4).



3.2.2 Menus

The following keys are defined, to go through the menus:

Home, End	Navigate in the menus
PgUp, PgDn	Navigate in the menus
↑,↓	Navigate in the menus
Esc	Go to the previous menu
↵	Select highlighted item

Items also can be selected by typing the character indicating the item.

3.2.3 Checklists

Checklist are displayed for instance at the next items:
Input - Flow - Boundary conditions
Input - Flow Structure control
Input - Quality - External Variables
Output - Define routes

A checklist is used to select one or more items from a list. A checkmark (√) in front of an item indicates the selection of the item. A checklist knows two modes, the command mode and the selecting mode.

In both modes the available commands are displayed in the status bar at the bottom of the screen. The commands available are listed below:

Command mode	Select mode
↵	Accepting actual choice
P	Print selected items
G	Make a graphic representation of the selected items
N*	Erase all defined data and create new
A*	Add new items
S,↑,↓	Go to select mode
PgUp, PgDn	Navigate in the checklist
Home, End	Navigate in the checklist
Esc	Return to previous menu
<space>	Select or de-select the indicated item
A	Select all items in the checklist
N	De-select all items in the checklist
PgUp ,PgDn	Navigate in the checklist
Home ,End	Navigate in the checklist
↑,↓	Navigate through the checklist
↵	Return to the command mode

* These commands are only available when the checklist is followed by one or several tables.

3.2.4 Tables

3.2.4.1 Command/edit mode

Tables are used to enter the actual data.

A table can be used in two modes; the command mode and the edit mode. The available commands are listed below:

Command mode	Edit mode	
Esc		Return to the previous menu
↑,←,E		Go to the Edit mode
P		Print the displayed table
G		Make a graphic representation of the related data
D		Enter the default values
V		Look at several variables with their description and defaults
PgUp, PgDn		Navigate in the table
Home, End		Navigate in the table
↵		Go to the following table/Return to the menu
	Esc	Go to the Commanding mode
	←, ↑, →, ↓	Navigate in the table
	PgUp, PgDn	Navigate in the table
	Home, End	Navigate in the table

=	Enter the previous value
Ins	Insert a new line of data
Del	Delete a existing line of data
↵	Accept the displayed value

The edit mode enables the user to enter the values into the table. When the highlighted item has several selection options, the program will show a pick list on the right side of the screen. An item also can be selected by typing the name of the variable to be selected.

The available commands in the edit mode are listed below:

Table	Pick list	
↵		Enter the actual highlighted value from the pick list (indicated with *)
Tab,→		Go to the pick list
↑, ↓		Navigate in the table
PgUp, PgDn		Navigate in the table
Home, End		Navigate in the table
	Tab,	Return to table
	↵	Select the indicated item, go to the table and enter the selected item
	↑, ↓	Navigate in the pick list
	PgUp, PgDn	Navigate in the pick list
	Home ,End	Navigate in the pick list

3.2.4.2 Default values

This command can be applied in the next menu items:
Input - Flow - Network - Nodes
Input - Flow - Network - Sections
Input - Flow - Initial condition - Levels
Input - Flow - Initial condition - Discharges
Input - Quality - Initial condition

Entering a complete table or a part of a table with defaults values is possible. After selecting D(efault), when indicated, a popup window appears. After entering the popup window the next commands are available:

Esc	Erase the popup window
←, ↑	Navigate through the popup window
↵	Fill the table with specified default values.

An example of a default popup screen is given below:

Change	Defaults				
(Y/N):	Yes	Yes	Yes	Yes	Etc.
Value:	1.000	2.360	0.0000	0.000	

When a default popup screen appears, it is filled with defaults values for the specified items by the program. The user can change the default values by using the two next actions:

(Y/N)	Enables to turn the default option on or off for the specified items on the main screen.
Value	For those items the default value should be used, the user can enter the desired value.

After completing the default popup screen, the default values will be entered in the table.

3.2.4.3 Time series

The use of time series can be found in the following menu items:
Input - Flow - Boundary condition
Input - Flow - Structure control - Continues
Input - Quality - Boundary condition
Input - Quality - External variables
Input - Measured data

The following possibilities are available to enter time dependent functions:

Constant
: The input variable is constant in time.

Fourier series
: The input variable can be formulated as a Fourier series.

$$y_t = y_0 + \sum_{k=1}^{N} y_k \cos(k\omega t - \Phi_k)$$

$$\omega = \frac{360}{P_1}$$

y_t	Value at time t.
y_0	Mean value.
y_k	Amplitude of k^{th} component.
k	Component number ($1 \leq k \leq 8$).
N	Number of components (≤ 8).
P_1	Cycle first component.
Φ_k	Phase of kth component (degrees).

Time series
: An equidistant time series which covers the entire simulation period plus one "Time step Quality".

"Start data"	"start of calculation" is given as default
"Start time"	
"number of values"	the minimum number to cover the whole calculation plus one "Time step Quality".
"time"	format: yymmdd hhmm

Non eq. series
: A non equidistant time series needs to cover the entire simulation period. Numerical values should be entered for date, time and actual value. The values used by DUFLOW will be calculated by means of linear interpolation.

Import file
: The input data will be read from an external ASCII file. The file name must be entered (including extension). The path where the file is stored must be entered. Two type of data files can be read:

Time series	A record of the data file contains only one variable value. The file itself may be of a free format, values can be separated by a blank, [Tab] or [Return]. After importing the "start date", "Start time" and "interval"

have to be specified. So these items must not be stored at the external file. The default values are equal to default format in calculations definition (see paragraph 3.4.1.1).

Non equidistant time series:

A record of the data file must contain date, time and the actual value. It is strongly recommended to use the default otherwise the chance of errors is large.
During the import the data will be written to a DUFLOW input file. When later on changes have been made in the ascii file the DUFLOW input file will not be updated.

3.2.4.4 Use of "Print" and "Graph" commands

The "Print" command is available at every table in the program.

The "Graph" command can be applied at the next items:
Input - Flow - Network - Definition
Input - Flow - Boundary conditions (checklist and individual table)
Input - Quality - Boundary conditions (checklist and individual table)
Input - Quality - External variables (checklist and individual table)
Input - Measured data - Individual table

The commands P(rint) and G(raph) enable the user to direct the output to a printer, plotter or disk. If in "Hardware configuration" (see paragraph 3.9.1) no printer or plotter is defined, DUFLOW will send the output to disk.

Print"	P	= directly to printer.
	F	= directly to file, the program prompts for a file name.
Graph"	P	= graph is send to the connected plotter.
	F	= graph is written to a HPGL file, the program will prompt for a file name (without extension).
	S	= graph is shown on screen.
	←, ↑	= enables to edit the layout of the graph.

3.3 MASTER MENU

The master menu is displayed at the start of the program and after returning from the accessible submenus. If an fatal error occurs, the control will also return to master menu.

At the top of screen the current data directory and the name of the project are displayed. The menu is displayed below:

```
DUFLOW  2.xx                Master Menu
Data directory : C:\DUFLOW\EXAMPLES\
Project        : TEST


                        MASTER MENU

                         I  Input
                         C  Calculations
                         O  Output

                         F  File names specification
                         M  quality Model development
                         S  Setup
                         Q  Quit
```

I=Input Starts the input module and displays "menu input" (see paragraph 3.4).

C=Calculations
Displays menu calculations (see paragraph 3.5).

O=Output Starts the output module and displays "menu output" (see paragraph 3.6).
Enables to make graphs and tables of the results and measured data, which are stored in the result files (*.res, *.rek and *.msr).

F=File names specification
Defines the name of data directory, project file, input and result files (see paragraph 3.7).

M=quality Model development
In this part the user should supply the process descriptions. These are stored in the model file (*.mod). After compilation a *.mob file is created which will be used by the program (see paragraph 3.8).

S=Setup Enter information concerning the hardware configuration, colours and limits to data size (see paragraph 3.9).

Q=Quit Terminates the DUFLOW program and returns to MS-DOS command level.

3.4 Menu INPUT

To edit or create a input data set. For creating a new input data set, start with the definition of a project and file names using "File names specification" in "master menu" (see paragraph 3.7) and continue with the control, flow and quality related data using options "control data", "flow data" and "quality data" from the input menu. For editing, the files in the defined project can be selected.

```
DUFLOW  2.xx                          Input - Menu
Data directory : C:\DUFLOW\EXAMPLES\
Project        : EUTROF1


                        MENU INPUT

                        C  Control data
                        F  Flow data
                        Q  Quality data

                        S  Save as...
                        D  measured Data

                        M  Master Menu
```

"C=Control data"

Control data determine how the calculation will be carried out and which results will be preserved for output (see paragraph 3.4.1).

"F=Flow data"

All flow related data like the network description, nodal points, sections, initial conditions, boundary conditions and structure control should be entered using this submenu (see paragraph 3.4.2).

"Q=Quality data"

All quality related data like the external variables, parameters, initial and boundary conditions should be entered using this submenu (see paragraph 3.4.3).

"S=Save as ..."

Special function to store input files (see paragraph 3.4.4).

"D=measured Data"

Enables the input of measured data for state variables and quality intermediate results defined as function identifier in DUPROL (see paragraph 3.4.5).

3.4.1 CONTROL DATA

```
DUFLOW  2.xx                    Input - Control data - Menu
Data directory : C:\DUFLOW\EXAMPLES\      Control data : RIEUT1.CTR
Project        : EUTROF1

                          MENU INPUT - CONTROL DATA

                          C  Calculation definition
                          L  Locations for output
                          Q  Quality variables for output
                          S  Special control data

                          M  Menu input
```

"C=Calculation definition"

Modify data controlling the computational module (see paragraph 3.4.1.1).

"L=Locations for output"

Selects the locations for which output will be written to the result files (see paragraph 3.4.1.2).

"Q=Quality variables for output"

The program will show the checklist with defined state quality variables and intermediate results of function identifiers. The available commands (see paragraph 3.4.1.3) can be used to control the desired output.

"S=Special control data"

In this submenu additional control features are available (see paragraph 3.4.1.4).

3.4.1.1 CALCULATION DEFINITION

In the table (below) the general calculation data have to be entered.

```
DUFLOW  2.xx                  Input - Control data - Calc. definition
CALCULATION DEFINITION

IDENTIFICATION TEXT: EXAMPLE
TIMES                       ¦ yymmdd  ¦ hhmm  ¦
  Start of computation      : 910701    0000
  Start of output           : 910701    0000
  End of computation        : 910703    0000
  Time step Flow            :           0010        # Time steps   =   288
  Timestep Quality          :           0010        # Time steps   =   288
 Output interval            :           0010        # Output steps =   288

HYDRAULIC CALCULATION-------------------------
  Resistance formula        : de Chezy
  Calc. of advection term   : Total
  Extra iteration           : No
  Theta           (0.5-1.0): 0.55
QUALITY CALCULATION---------------------------
  Decouple                  : Yes
  Theta           (0.0-1.0): 0.55
```

"Identification text"

For identification of a run, one line of text can be specified.

"Start of computation"

The start date and time of the simulation.

Format date: yymmdd =	year month day (each taking two positions adjacent to each other)
Format time: hhmm =	hour minute (each taking two positions adjacent to each other)

"Start of output"

The start date and time of writing data to the result file. This enables the user to skip the output of the first part of the simulation. This can be convenient if the initial conditions do not represent the physical state of the system at the start of the simulation.

Format date:	yymmdd = year month day
Format time:	hhmm = hour minute
Default:	equal to "Start of computation"

"End of computation"

The end date and time of the simulation.

Format date:	yymmdd = year month day
Format time:	hhmm = hour minute
Default:	equal to "Start of output"

"Time step Flow"

Time interval used for calculation in the hydraulic part. Chapter 2 gives suggestions for the choice of the hydraulic time step.

Format time:	hhmm = hour minute
Default:	0010 (10 minutes)

"Time step Quality"

Time interval for the quality part of the model. Chapter 2 gives suggestions for the choice of the quality time step. The "Time step Quality" is rounded to the nearest multiple of "Time step Flow".

Format time:	hhmm = hour minute
Default:	0010 (10 minutes)

"Output interval"

Interval for writing data to the result files, starting at "Start of output". The "Output interval" is rounded to the nearest multiple of "Time step quality".

Format time:	hhmm = hour minute
Default:	equal to "Time step Flow" if only flow is calculated and equal to "Time step Quality" if the quality model is also used.

"Resistance formula"

The channel friction (see paragraph 3.4.2.1.3) can be calculated using

- the formula of De Chézy. The "Resistance" coefficient C in the definition of the sections is from the basic formula:
  $v = C.\sqrt{R.I}$

- formula of Manning. The "Resistance" is the Manning coefficient k (1/n) from the basic formula $v=k*R^{2/3}*I^{1/2}$. In the actual calculation $C=k*R^{1/6}$ is substituted in the formula of De Chézy. The value of C is calculated for each time step during the simulation.
  Default: De Chézy

"Calc. of advection term"

Option T (Total) includes the so-called Froude term (see paragraph 2.2.1). Selecting D ('Damped') the Froude term is also taken into account, but its absolute value will not exceed the friction term. Using N (No) the Froude term is neglected.

"Extra iteration"

At each time step the level dependent parameters (like flow area, storage area and hydraulic radius) are calculated from the situation at the previous time. These values can be adjusted using the 'new' values computed for the actual time using an extra iteration (see paragraph 2.2.2).
Default: Yes

"Theta (0.5 - 1.0)"

The factor controlling numerical damping (see paragraph 2.2.2).
Default: 0.55

"Decouple"

Option N (No) considers dispersion at both sides of a node. Option Y (Yes), only dispersion in forward direction is taken into account (see chapter 2). Decoupling only takes place at those nodes, where a discharge is located (also see paragraph 2.3).

"Theta (0.0 - 1.0)"

The factor controlling the numerical solution of the advection and dispersion equation (see paragraph 2.5). The value zero ($\theta = 0$) is for mathematical reasons not allowed.
Default: 0.55

3.4.1.2 LOCATIONS FOR OUTPUT

In the table (below) the locations for output have to be entered.

```
DUFLOW  2.xx                          Input - Control data - Locations

MONITORING OUTPUT
 Selected sections/struct. :


OUTPUT DEFINITION - LOCATIONS
 Selected sections/struct. :
```

"Monitoring output"

This option enables the output of various variables for sections and structures. Up to eight "Structures" or "Sections" can be selected. The results will be written to *.mon file.
Various variables for channel sections are:

H1, H2	Levels at begin and end (m)
Q1, Q2	Discharge(s) at begin and end (m^3/s)
A1, A2	Flow profile area at begin and end (m^2)
B1, B2	Storage area at begin and end (m^2)
V	Velocity in the middel of the section
WIDTH	Storage width
R*C2	Resistance term (F=(|Q|* v)/($R*C^2$))

For all structures:

H1, H2	Levels at begin and end (m)
Q	Discharge at structure (m^3/s)

For weirs and culvert only:

V	Velocity in the structure
A	Flow area at the structure
MU	Actual value of MU
Formula	Number of the used formula (see paragraph 2.2.4.2)

For siphons only:

V	Velocity in the structure
MU	Actual value of MU
FULL/NOT	State of the siphon: 1=full, 0=empty.

For pumps only:

ON/OFF	State of the pump: 1=on, 0=off

"Output locations"

Output is written to the flow result and quality result output files (*.res and *.rek). In this menu the locations, for which output has to be written to the file, can be selected. Only the selected data can be used for representation in tables and graphs in the "Output" (see paragraph 3.6). Only the existing sections/structures can be selected. Only if there are more than 8 sections/structures All can be selected.

3.4.1.3 QUALITY VARIABLES FOR OUTPUT

If a process description file is edited and compiled correctly in "quality Model development" (see paragraph 3.8), variables can be selected for output. From the displayed checklist state variables and intermediate results defined as function identifiers in the DUPROL process description file can be selected for output.

For the available commands see paragraph 3.2.3.

3.4.1.4 SPECIAL CONTROL DATA

```
DUFLOW  2.xx                          Input - Control data - Special

SPECIAL CONTROL DATA
Create intermediate flow file          : Yes
Alpha (corr. for velocity distribution):  1.000
Minimum #timesteps between triggers    :    3
```

"create intermediate flow file"

A dump file will be generated during simulation using option Flow (see paragraph 3.5). The intermediate flow result file (*.dmp) will be used to simulate transport if option "quality" is selected in the "menu calculations". In this *.dmp file information about volumes, flows etc. is stored, which is read during quality simulation. This enables the user to perform a number of quality simulations using the same hydraulic conditions.
Default: No

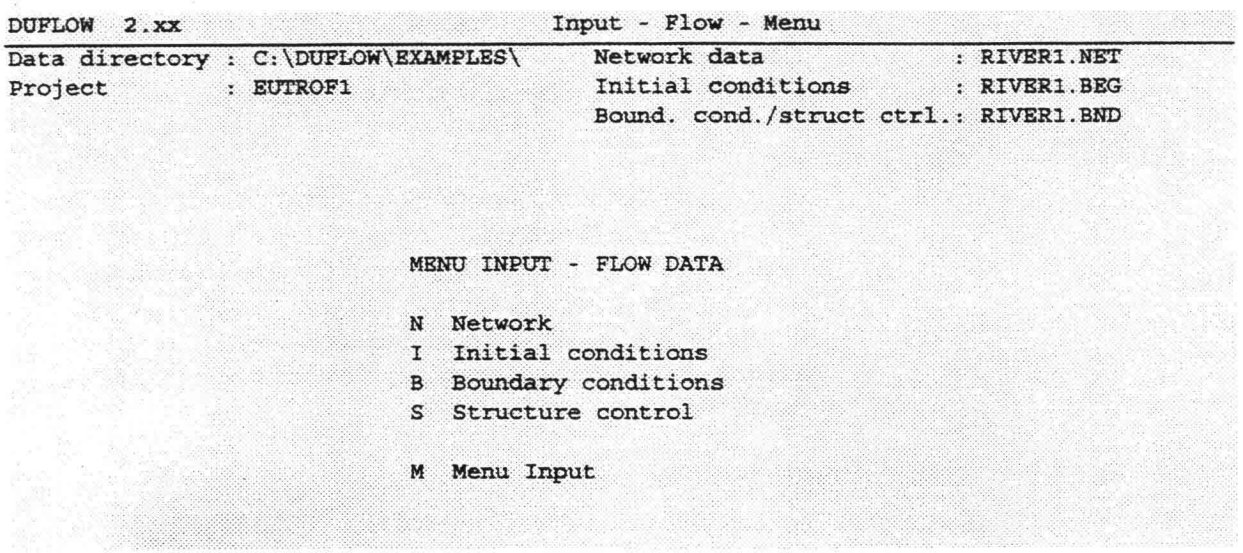
"Alpha (corr. for velocity distribution)"

Represents the influence of the velocity distribution over the cross-section, see paragraph 2.2.2.
Default: 1

"Minimum number of time steps between triggers"

After a trigger condition is affected some shocks can occur due to a sudden change in the structure operation. To prevent the next trigger from reacting to these shocks, the check for this next trigger is omitted during a specified number of time steps.
Default: 3

3.4.2 FLOW DATA



"N=Network"

Jumps to "network" (see paragraph 3.4.2.1).

"I=Initial conditions"

Jumps to "initial conditions" (see paragraph 3.4.2.2, 3.4.3.1).

"B=Boundary conditions"

Jumps to "boundary conditions" (see paragraph 3.4.2.3).

"S=Structure control"

Jumps to "structure control" (see paragraph 3.4.2.4).

3.4.2.1 NETWORK

In this menu the network definition has to be entered. The "menu network" will store the information in separated two data files (*.nod, *.net). These files are indicated at the top of the screen.

Selection items are:

```
DUFLOW  2.xx                          Input - Flow - Network - Menu
Data directory : C:\DUFLOW\EXAMPLES\     Network data              : RIVER1.NET
Project        : EUTROF1

                        MENU INPUT - FLOW DATA - NETWORK

                        D  network Definition
                        N  Nodes
                        S  Sections
                        C  Cross-sections
                        U  strUctures

                        M  Menu input - flow data
```

"D=network Definition"

Jumps to "network definition" (see paragraph 3.4.2.1.1).

"N=Nodes"

Jumps to "nodes" (see paragraph 3.4.2.1.2).

"S=Sections"

Jumps to "sections" (see paragraph 3.4.2.1.3).

"C=Cross-sections"

Jumps to "cross-sections" (see paragraph 3.4.2.1.4).

"U=strUctures"

Jumps to "structures" (see paragraph 3.4.2.1.5).

3.4.2.1.1 NETWORK DEFINITION

In this table (below) the lay out of the network has to be entered. For each section begin and end node must be given.

```
DUFLOW  2.xx                          Input - Flow - Network - Definition
NETWORK DEFINITION                    Press TAB to switch to node renumbering
 Section/   Begin     End
 Structure  Node      Node

                                                Orientation of network:
                                               0.00  ° from N to Y axis
```

The network may consist up to 448 nodes. For the sections numbers the numbers 1 - 300 are available. For the structures the numbers S1 - S148 are available.

A warning will be given when the network is inconsistent.

The option "Tab" displayed on the screen, enables the renumbering of the network. The user should keep in mind that only the nodes are renumbered and not the section numbers.

The options are displayed at the bottom of the screen, for further explanation see paragraph 3.2.4.

"Orientation"

Enables the user to change the orientation of the entire network. The orientation is measured clockwise from North towards the Y axis in degrees.

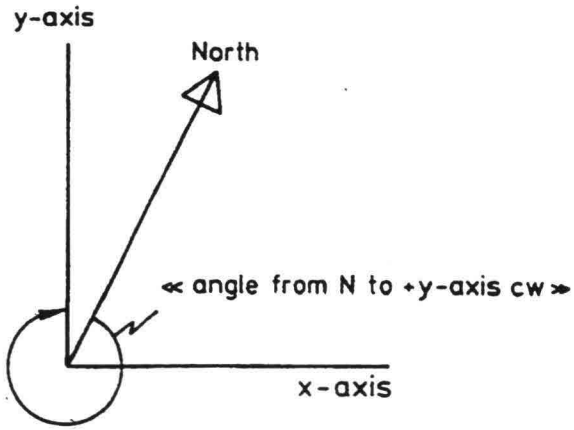


Figure 3.1 Orientation of the network

3.4.2.1.2 NODES

Enter all data related to a node.

```
DUFLOW  2.xx                          Input - Flow - Network - Nodes
NODES general and flow related parameters
| Node | X-Coord. (m) | Y-Coord. (m) | Catchment Area (ha) | Runoff Factor |
```

"X - co-ordinate"
"Y - co-ordinate"

Co-ordinates in the horizontal plane of a node relative to the origin.
A plot of the network can only be made if the co-ordinates are available. The input of the co-ordinates is optional. If entered, the length of the section is calculated using these co-ordinates. If not entered, the user must supply the length of the sections in "sections" (see paragraph 3.4.2.1.3). If the nodes co-ordinates are changed during the input session, to obtain the default length of the section, press 0 at that particular section.
Unit: meter

"Catchment area"

To be used in combination with the "rain" Boundary condition.
The "catchment area" is assumed to drain into the channel system at that node.
Unit: hectare

"Run-off factor"

The fraction of the total precipitation that discharges to the node.
Unit: dimensionless
Default: 1.0

3.4.2.1.3 SECTIONS

In this menu all section related data have to be entered.

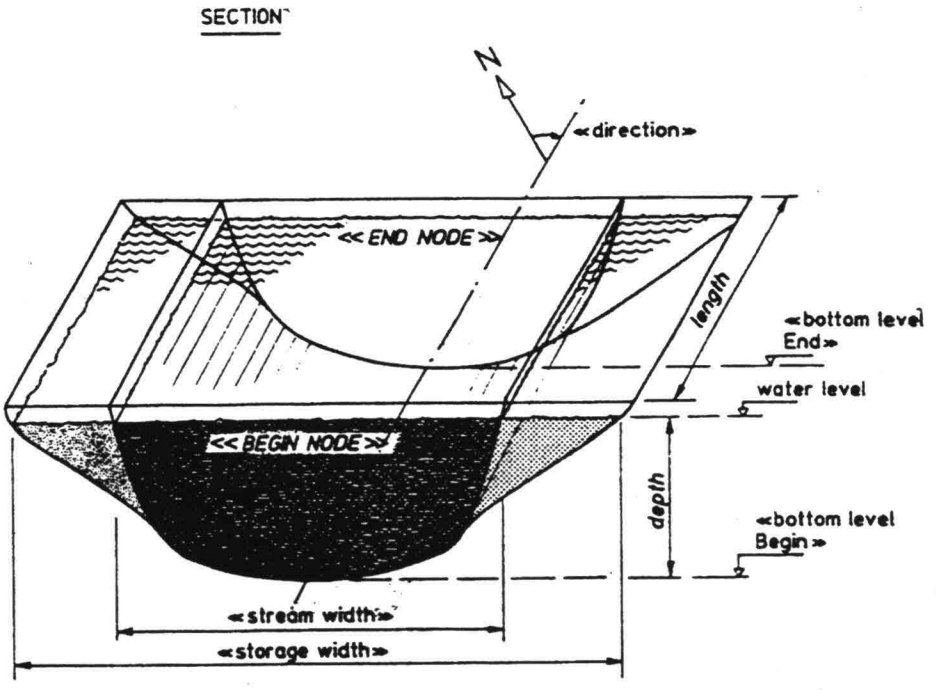


Figure 3.2 Section

DUFLOW 2.xx Input - Flow - Network - Sections

SECTIONS general and flow related parameters

Section	Length	Direction °CW fr.N	Bottom Level (m) Begin	Bottom Level (m) End	Resistance (C or k) Pos. dir.	Resistance (C or k) Neg. dir.	Windconv. (10^-6)

"Length" The length of the section.

Default: The distance between "Begin node" and "End node", calculated from the co-ordinates.

Unit: meter

"Direction" Angle measured clockwise from the North to the positive section direction (from "Begin" to "End").

Default: Direction from "Begin node" to "End node" calculated from the node co-ordinates.

Unit: degrees (360°)

"Bottom level"

Level of the lowest point of the cross-section with respect to the reference level.

Default: The last specified bottom level in that same node.

Unit: meter (positive is above reference level)

"Resistance" If in "control data" De Chézy was specified then "Resistance" is the Chézy coefficient. If Manning was specified "Resistance" is

the coefficient K (=1/n) in the formula of Manning: $v=K*R^{2/3}*I^{1/2}$. Distinction can be made between resistance experienced by flow in positive direction and by flow in negative direction.

Default: In positive directions the value at the former section; in negative direction the value for positive direction at the same section.

Unit: De Chézy: $m^{1/2}/s$
Manning: $m^{2/3}/s$

"Windconv" Wind conversion coefficient * 10^{-6}. Sheltering from the wind by dikes, buildings etc. can be simulated by specifying a lower value of "Windconv.".

Default: 3.6
Unit: 10^{-6}

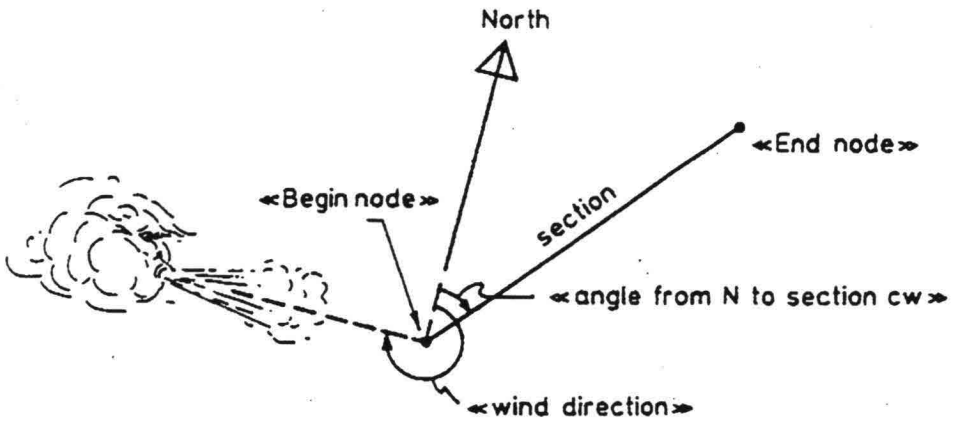


Figure 3.3 Definition of wind direction

3.4.2.1.4 CROSS-SECTIONS

This menu enables to enter all cross-section related input.

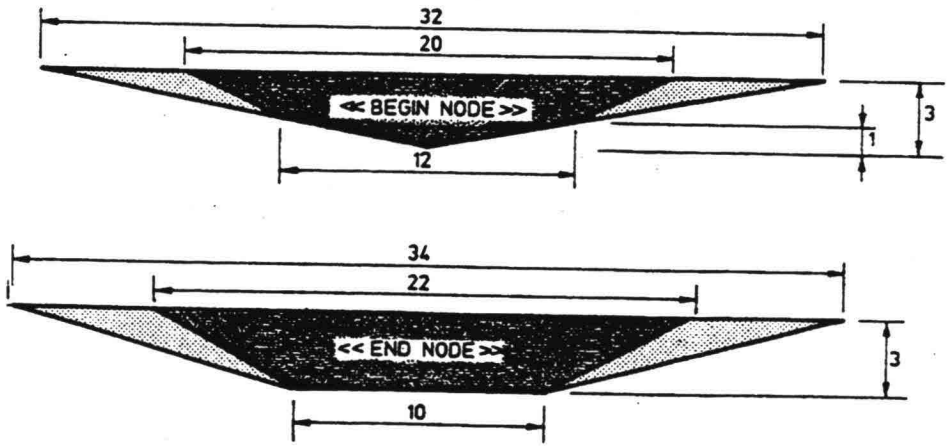


Figure 3.4 Cross Section

The specification is shown in the table below:

DUFLOW 2.xx Input - Flow - Network - Cross-sections

CROSS-SECTION Profile SECTION 1

Depth to bott.(m)	Flow width (m) at begin	Flow width (m) at end	Storage Width (m) at begin	Storage Width (m) at end
0.00	0.00	10.00	0.00	10.00
1.00	12.00	12.00	12.00	18.00
3.00	20.00	22.00	32.00	34.00
0.00				

"Cross-section, Profile"

The profile of the cross-sections is specified by the Width-of-flow profile and the Width-of-storage profile at the "begin node" and at the "end node". These profiles may be composed of up to 15 depths, where the first line always gives the bottom width. The table is terminated by entering a depth of 0 meter.
Note that the width above the last specified depth is assumed to be equal to this last width.

Defaults: The width value at the former depth, but for "width at end node", if greater, the value at begin node and for "storage width", if greater, the average flow profile width.

Units: meters

In this table two additional options are available in command mode:

"Copy" Enables duplication of an earlier defined cross-section by replying to "Give number of section to copy from". The cross-sectional data displayed on the screen will be replaced.

"Hydraulic data"

Gives an option to overrule the implicitly calculated hydraulic specification of the cross-section.
In normal cases the program will calculate the actual flow profile areas(A) and the hydraulic radius(R) from the specified profile:

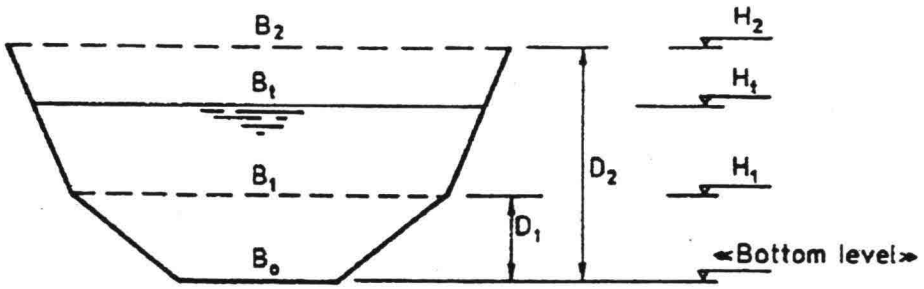


Figure 3.5 Calculation of hydraulic data

The "Resistance" given in the table "Sections" is valid at every depth.
In the "Hydraulic data" table the user can specify these parameters as functions of the depth.

DUFLOW 2.xx Input - Flow - Network - Cross-sections

CROSS-SECTION Hydraulic data SECTION 1

Depth to bott.(m)	Flow Area (m2) begin	end	Hydraulic Radius(m)	Resistance (C or k) pos. dir.	neg. dir.
0.00	0.00	0.00	0.00	30.00	30.00
1.00	6.00	11.00	0.68	30.00	30.00
3.00	38.00	45.00	1.85	30.00	30.00

If any change is made in the columns of "Flow Area", "Hydraulic radius" or "Resistance" then all implicitly computed values of that item will be overwritten. Press "Defaults" in the command mode to reset.
This table contains an extra "Print" option, "All" will print all selected cross-sections.

For further explanation about options see paragraph 3.2.4.

3.4.2.1.5 STRUCTURES

"Type" Various types of structures can be specified:

"1 = Overflow" A short weir with free overflow.

"2 = Underflow" A short weir with 'forced' underflow.

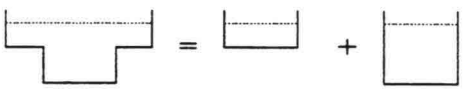
"3 = Culvert" A weir with additional friction

"4 = Siphon" A circular pipe to carry water over obstacles

"5 = Pump" A pumping station (can switch on and off)

"6 = Elliptic culvert" A weir with additional friction and an elliptic cross-section

Weirs and culverts have rectangular cross-sections, except for the elliptic culvert. Structures with a more complicated cross-section can be simulated as two or more parallel structures with the same begin and end node.



"Width" (Options 1, 2, 3). The width of the rectangular flow opening.
Unit: m

"Sill Level" (Options 1, 2, 3). The level of the bottom of the rectangular flow opening.
(Option 6). The level of the bottom within the elliptic culvert. The part of the elliptic culvert cross-section below the level that contain water and is a part of the flow profile.
Unit: m

"Gate level" (Options 2, 3). The level of the top of the rectangular flow opening.
Unit: m

"Length" (Options 3, 4, 6). The length of a culvert or siphon.
Unit: m

"Diameter" (Option 4) The diameter of the siphon.
Unit: m

"Mu...." (Options 1, 2, 3, 4, 6). Mu is a correction factor for additional hydraulic effects like contraction, internal friction etc.
"Mu pos. dir": Mu for flow in the positive direction (from begin to end).
"Mu neg. dir": Same in negative direction.
Default: 1
Unit: dimensionless

The table in command mode contains an extra "Print" option, "All" will print all selected structures.

"Mu submerged flow"
(Options 1, 2, 3, 6) Mu for the submerged flow situations.
Default: 0.83 * mu-free surface.
Unit: dimensionless

"Chézy" (Options 3, 4, 6) The Chézy coefficient for the friction in a culvert or a siphon. The friction in the structures is always calculated with the formula of Chézy.
Unit: $m^{1/2} s^{-1}$

"Capacity" (Option 5) Discharge through the pump from "begin node" to "end node".
Unit: m^3/s

"Start level" }
"Stop level" }
(Options 4, 5) Switching levels. The flow in a siphon starts if both levels at begin and end are above "start level" and stops if one of them sinks below its "stop level". For a pump "start level" and "stop level" are related to "begin node".
Unit: m

"Horizontal rad(ius)"
(Option 6) Half of the horizontal axes of the elliptic cross-section of the elliptic culvert.
unit: m

"Vertical radius"
(option 6) Half of the vertical axes of the elliptic culvert.
Unit: m

"Inside level" (option 6) The level of the lowest point of the elliptic cross-section.
Unit: m

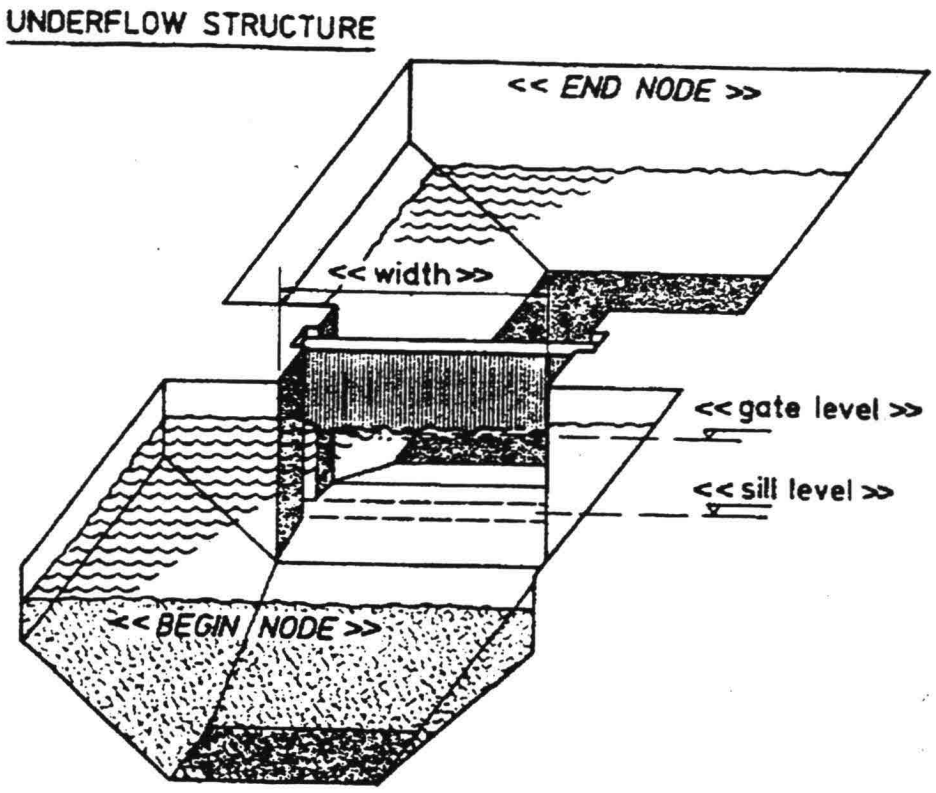


Figure 3.6a Weir with underflow

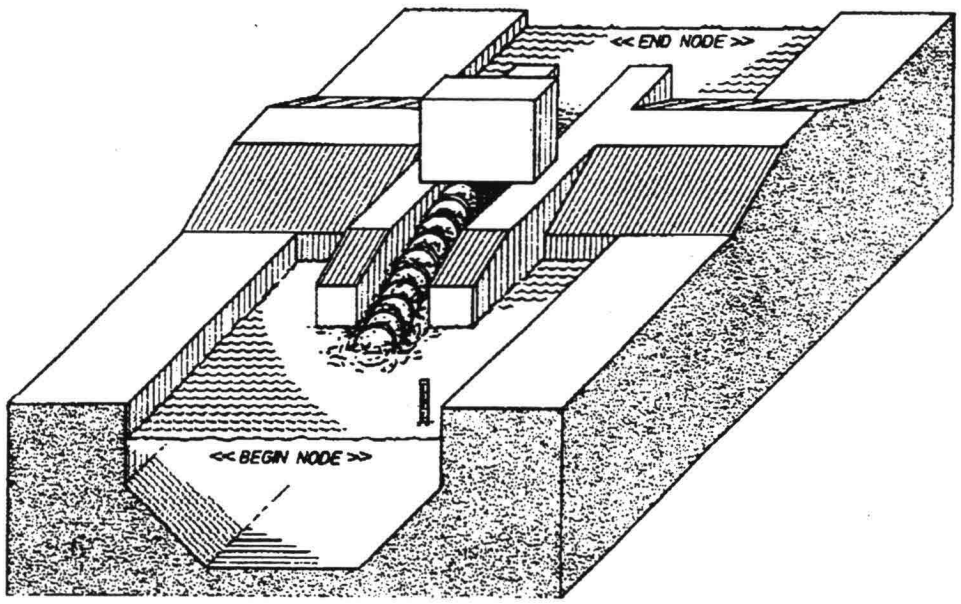


Figure 3.6b Pump

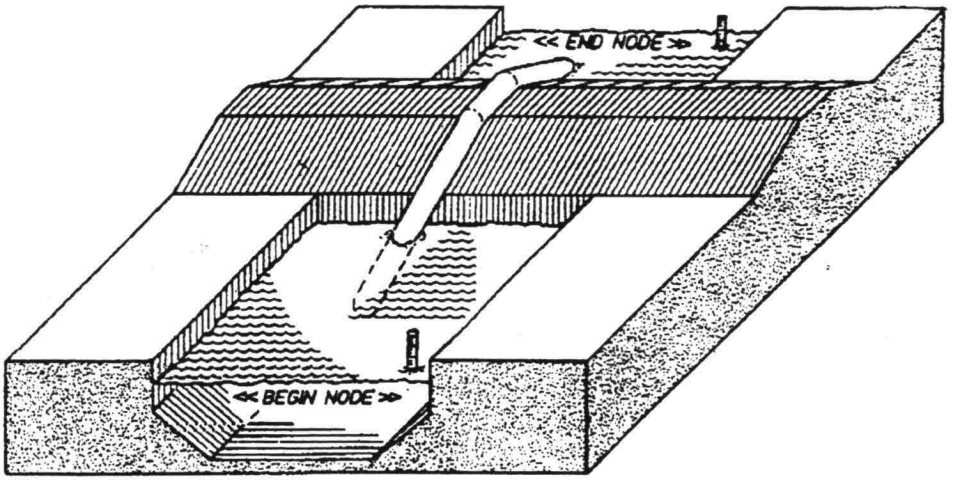


Figure 3.6c Siphon

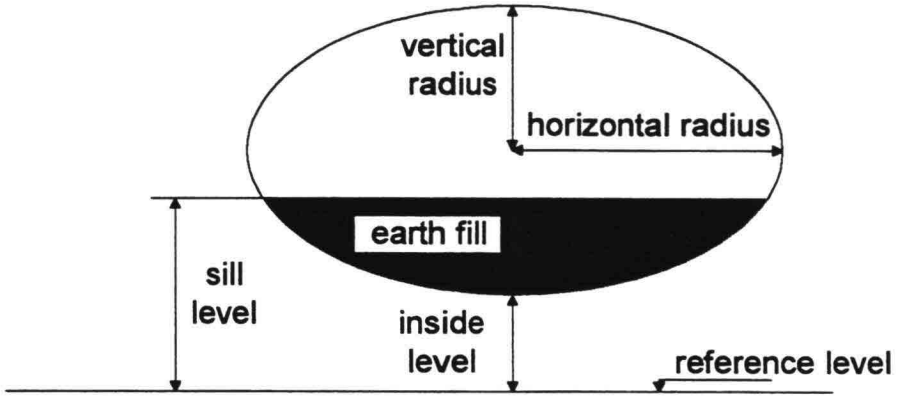


Figure 3.6d Elliptic Culvert

3.4.2.2 INITIAL CONDITIONS

```
DUFLOW  2.xx                          Input - Flow - Initial cond. - Menu
Data directory : C:\DUFLOW\EXAMPLES\     Initial conditions       : RIVER1.BEG
Project        : EUTROF1


                    MENU INPUT - FLOW DATA - INITIAL CONDITIONS

                    H  initial levels
                    Q  initial discharges

                    M  Menu input - flow data
```

"H=initial levels"

Specify the initial level at individual nodes.
Unit: meter (with respect to the reference level)
With the extra option "Defaults", the user enter values in two ways;

- option Levels enables to enter absolute levels with respect the reference level;
- the option Depths enables to enter a height above the specified bottom levels. The program will calculate the values with respect to the reference level.

"Q=initial discharges"

Specify the initial discharge at begin and end of each section. For structures one value is given.
Unit: m^3/s (positive values from begin node to end node).
The initial discharges are set to 0 m^3/s if no values are entered.

Additional options are available, for further explanation see paragraph 3.2.4.

3.4.2.3 BOUNDARY CONDITIONS

This table can be used to describe the conditions at the physical boundaries of the system. At these nodal points a water level or a flow boundary condition can be used. This menu is also used to enter discharges. These can be specified at each node. In the hydraulic part more than one discharge can be used at a nodal point. In the quality part however the number of discharges is restricted to one.

In this menu also some general flow related input variables can be supplied, like rain, wind velocity and wind direction.

If one or more boundary conditions already have been defined the programs will show the checklist in command mode. When no boundary conditions exist the program will immediately display the edit mode to enter the specification of the first boundary condition. For explanation of the commands see paragraph 3.2.3 and 3.2.4.

The following individual specifications can be entered in edit mode:

"Type" Next selection items are available:

"Level" Prescribed water level
Unit: meter

"Q add" Controlled additional inflow to the network, a withdrawal can be specified using a minus sign.
Unit: m^3/s .

"Q add. Hrel."
Level dependent additional inflow to the network. Up to 20 combinations of a level and corresponding discharge can be specified.
Unit: m^3/s .

"Rain" Precipitation, generating an additional inflow to the nodes. Is related to "Catchment area" and "Run-off factor" (see paragraph 3.4.2.1.2):
Qadd= R * 0.001 * A *10000 * c /(24 *3600)
where:
Qadd = Additional inflow to the node (m^3/s)
R = Precipitation intensity (mm/day)
A = "Catchment area" (ha)
c = "Run off factor"
Unit: mm/day

"Wind velocity"
Wind velocity, uniform over the whole network.
Unit: m/s.

"Wind direction"
Angle of wind direction, from North to origin and measured clockwise (90° means wind from the East). Uniform over the whole network.
Unit: degrees (between 0 and 360)

"None (erase)"
Erase the already existing boundary condition. Stop adding new ones.

"Node" A boundary condition can be valid for only one node.

"Condition number"
Number to identify additional inflow to the network, used when more than one additional inflow is defined, on a node. The number (integer) statet here (flow boundery conditions) must agree whit the number statet in quality boundery conditions.

Default: 0

"Type of function"
Boundary conditions can be specified in five ways:
"Constant"
"Fourier series"
"Time series"
"Non eq. series"
"Import file".

Additional explanation of the individual function see paragraph 3.2.4.3.

3.4.2.4 STRUCTURE CONTROL

If one or more structure operations already have been defined the program will show the checklist in command mode. When no structure operations exist the program will immediately display the edit mode to enter the specification of the first structure operation. For explanation of the commands see paragraph 3.2.3 and 3.2.4.

After selecting or entering new operations the following individual specifications can be entered:

"Type of operation"

"Continuous"

The operation will be executed during the entire period of the calculation.

"Operational parameter"

The parameter of the structure that will be affected by this structure operation. Note that earlier parameter values are overwritten.

"Structure number"

Specify the structure(s). Take care that the "operational parameter" has a meaning for the given structure(s) (e.g. do not change the sill level of a pump).

"Type of function"

Continuous operations can be specified in five ways:
"Constant"
"Fourier series"
"Time series"
"Non eq. series"
"Import file".
Additional explanation of the individual function see paragraph 3.2.4.3 .

"Trigger series"

A `scenario' that describes the variation of one parameter in one or more structures depending on actual hydraulic conditions, called "trigger conditions". A `scenario' may consists of more than one trigger series. The number of operations times and the number of structures per operation may not exceed 16. Each "structure operation" may have up to 99 "trigger conditions"

"Operational parameter"

The parameter of the structure that will be affected by this structure operation. Note that earlier parameter values are overwritten.

"Type of trigger condition"

"Time"

After the specified time has passed the next trigger is checked. If at that moment the new condition is met, it is activated immediately.

"H2 > H1+ ΔH"

"Operational parameter" is executed if the level at "End" Node is higher than the level at "Begin Node" plus an increment ΔH.

"H1 > H2+ ΔH"

"Operational parameter" is executed if the level at "Begin" Node is higher than the level at "End Node" plus an increment ΔH.

"Hnode > Htrig"

"Operational parameter" is executed if the level at a node somewhere in the network ("Trigger node") is higher than a specified level ("Trigger level").

"Hnode < Htrig"

"Operational parameter" is executed if the level at a node somewhere in the network ("Trigger node") is lower than a specified level ("Trigger level").

"Repeat other trigger"

A trigger condition which already has been specified can be repeated by entering its serial number.

"No more triggers"

The series of triggers in this structure operation is terminated.

"Structure number(s)"

Specify the structure(s). Take care that the "operational parameter" has a meaning for the given structure(s) (e.g. do not change the sill level of a pump).

"Type of function"

The way how the value of "operational parameter" will change when the trigger condition is met can be specified in two ways:

"Constant"

A new value is assigned to the "operational parameter" until the next condition is met.

"Time series"

The "operational parameter" will change according to the given time series. The first value will be assigned immediately after the condition is met. The last value will be kept until the next condition is met.

If the next trigger condition is met before the end of this time series then the new "action" overrules the 'old' one.

"Erase operation"

Erase the already defined operation.

3.4.3 QUALITY DATA

```
DUFLOW  2.xx                          Input - Quality - Menu
Data directory : C:\DUFLOW\EXAMPLES\     Initial conditions      : RIEUT1.BEK
Project        : EUTROF1                 Boundary conditions     : RIEUT1.BNK
Model          : EUTROF1.MOB             External variables      : RIEUT1.EXT
                                         Parameters              : RIEUT1.PRM


                    MENU INPUT - QUALITY DATA

                    I  Initial conditions
                    B  Boundary conditions
                    E  External variables
                    P  Parameters

                    M  Menu Input
```

All quality related input can be edited through this menu. Note that the selection of a set variables for output has to be done in "quality variables for output" (see paragraph 3.4.1.3).

"I=Initial conditions"

Jumps to "initial conditions" (see paragraph 3.4.2.2, 3.4.3.1).

"B=Boundary conditions"

Jumps to "boundary conditions" (See paragraph 3.4.3.2).

"E=External variables"

Jumps to "external variables" (See paragraph 3.4.3.3).

"P=Parameters"

Jumps to "parameters" (see paragraph 3.4.3.4).

3.4.3.1 INITIAL CONDITIONS

After entering "INITIAL CONDITIONS", the programs asks which nodes to select for editing. If all nodes have to be selected, press ↵.In both cases the program will show the checklist in command mode. The available commands are explained in paragraph 3.2.3. After selection of the state variables the user can enter the edit mode to enter the initial values at the nodes. The available commands for edit mode are explained in paragraph 3.2.4.

3.4.3.2 BOUNDARY CONDITIONS

This table can be used to supply the conditions at the physical boundaries of the system. If a flow or water level boundary condition is used in the hydraulic part of the model, a concentration boundary condition is assumed at the same nodal point. The mass entering the system at this point is calculated from Q*C, where Q is the additional flow entered at "boundary conditions - flow data" or the flow calculated over a water level boundary. If no concentration is specified at a flow boundary condition the concentrations of the flow entering the system is set equal to zero. In the hydraulic part only one discharge can be entered at a nodal point.

In this menu also the loads has to be supplied. The load is calculated from Q*C [mg/l * m^3/s]. In the quality model more than one additional load can be connected to a certain node. A load is not connected with any flow discharge on the same node.

After choosing the "boundary conditions" from the menu, the programs shows the checklist of already defined boundary conditions in command mode. When no boundary conditions exists, the program will start with the first condition in edit mode.

The available commands are explained in paragraph 3.2.3. After selection of the state variables the user can enter the edit mode to specify the boundary conditions. The available commands for edit mode are explained in paragraph 3.2.4.

In the edit mode the individual boundary condition can be specified by the following items:

"Type" "Concentration"
The "Variable" normally will be entered like a concentration type towards a node when both discharge and mass are not negligible.
"Load"
The "Variable " also can be entered like a load type towards a node when the discharge of water is negligible compared with the mass.
"None (erase)"
No specification, to end or erase the already existing specification.

"Variable" From the pick list, shown at the right side on the screen, the state variables can be selected. For commands see paragraph 3.2.4.1.

"Node(s)" A boundary condition can be valid for one node.

"Condition number"
Number to identify the boundery condition. The number statet here (quality boundary conditions) must agree with the number statet in flow boundary conditions.

"Type of function"
Boundary conditions can be specified in five ways:
"Constant"
"Fourier series"
"Time series"
"Non eq. series"
"Import file"

Additional explanation of the individual function see paragraph 3.2.4.3.

3.4.3.3 EXTERNAL VARIABLES

The program shows directly the checklist of all defined XT external variables (space and time dependent) in the process description file (see paragraph 3.8). The dispersion coefficient is also considered to be an external variable although it

is not declared in the process description file. The commands for command mode are explained in paragraph 3.2.3.

In the edit mode the individual external variable can be specified (for commands see paragraph 3.2.4), by the following items:

"Type" From the pick list shown at the right side on the screen the external variables can be selected:

- the XT declared variable in the process description file
- d (= dispersion coefficient). For commands see paragraph 3.2.4.1.

"Node(s)" An external variable can be valid for more than one node. More nodes can be specified as separated by colons(,) or a hyphen (-) for a whole range (e.g. 3,5-7,10). If the user wants to select all nodes, then enter All.

"Type of function"
External variables can be specified in five ways:
"Constant"
"Fourier series"
"Time series"
"Non eq. series"
"Import file"
Additional explanation of the individual function see paragraph 3.2.4.3.

If an external variable is not specified the default value given in the process description file is used. Without specification, the dispersion coefficient is set equal to zero.

3.4.3.4 PARAMETERS

The program directly shows the table of parameters declared in the process description file. The available commands are explained in paragraph 3.2.4.1. The first two lines of the parameter table are shown below:

DUFLOW 2.xx Input - Quality - Parameters

MODEL PARAMETERS

Name	Description	Default	VALUE	Unit
achlc	chlorophyl to carbon ratio	30.000	30.000	ug-hl/mg-
anc	nitrogen to carbon ratio	0.25000	0.10000	mg-n/mg-c
aoc	oxygen to carbon ratio	2.670	2.670	mg-o2/mg-c
apc	phosphorus to carbon ratio	0.02500	0.01000	mg-p/mg-c
e0	background extinction	1.000	2.000	1/m
ealg	specific extinction chlorophyll	0.01600	0.01700	ug-chl/l,m

"Default" These default values are located in the process description file (*.mob). Using this option will fill the entire table with the default values.

"Edit" The only item that can be edited is "VALUES". New values will overwrite the old value, for this parameter.
The values of the variables under "VALUES" are used in "CALCULATIONS".

3.4.4 SAVE AS...

This menu shows the status of the current project.

Leaving an input menu if changes have been made, the user is prompted "Save to file? (Y?N)". If Yes is selected the changed data are written to the input file displayed at the right top of the screen. If No is selected the modified data are still in memory and can be saved using "save as ...". This option gives the opportunity to write the data another file name.

DUFLOW 2.xx Input - Save as ...

AVAILABLE DATA:

Type of data	Loaded	Modified	Filename	Save	Save as
Control data	Yes	-	RIEUT1.CTR	No	RIEUT1
Flow					
Network data	Yes	Yes	RIVER1.NET.NOD	Yes	RIVER1
Initial conditions	Yes	-	RIVER1.BEG	No	RIVER1
Bound.Cond./struct ctrl.	-	-	RIVER1.BND	-	-
Quality					
Initial conditions	-	-	RIEUT1.BEK	-	-
Boundary conditions	-	-	RIEUT1.BNK	-	-
External variables	-	-	RIEUT1.EXT	-	-
Parameters	Yes	-	RIEUT1.PRM	No	RIEUT1

"loaded" Indicates whether the data are present in memory. loaded.

"modified" Indicates whether the data have been changed and not have been saved yet, leaving the particular input menu.

"Save" In this column the user can enter if the modified should be saved.

"Save as" If desired the name of the file, where the data are written to can be changed.

When the data are changed and the user did leave the menu answering Yes, the data will be written in the current file. Nevertheless is it still possible to write the data to another data file.

3.4.5 MEASURED DATA

This menu enables to enter measured data. This option can be used for state variables (e.g. Q, H, u, C etc.) and function identifiers in the process description file. This enables to compare the results with measured data in the output module (see paragraph 3.6). The entered data will be stored in a *.msr file.

The program will give a checklist of the already existing data series in the file. Otherwise the program will start with the new measurement series 1.

The measurement series can be specified in the following way:

"Type" Following items are possible.

"Level" Water level.
Unit: meter.

"Discharge" Flow in section.
Unit m^3/s.

"Velocity" Water velocity in section.
Unit m/s.

"Concentration"	State variables defined in model file.
"Load"	Load in section calculated as Q*C.
"Function"	Function identifiers declared in the model file.
"None"	No selection is entered./Erase an existing series. /Stop adding new ones.

"Variables" Select variable for which measured data will be entered. The item will only appear when "Concentration", "Load" or "Function" is selected. Selection from the pick list can be done with the name of variable or pressing "Tab" for entering the pick list right on the screen (see paragraph 3.2.4.1).

"Node(s)"
"Section" Measured data can be used for more than one node, section or structure.

"Type of function"
Several types of function can be selected (see paragraph 3.2.4.3).
"Constant"
"Fourier series"
"Time series"
"Non eq. series"
"Import file"

3.5 Menu CALCULATIONS

This part of "master menu" performs the calculations in the way specified in "menu control data". Below the screen is shown, when the item has been entered from "master menu".

```
DUFLOW  2.xx                          Calculations
Data directory : C:\DUFLOW\EXAMPLES\
Project        : EUTROF1


                        MENU CALCULATIONS

                        A  All
                        F  Flow
                        Q  Quality
                        B  Box

                        M  Master Menu
```

"A=All" Flow and quality are calculated simultaneous.

"F=Flow" Flow is calculated.

"Q=Quality" This option only can be used if an intermediate flow result file (*.dmp) was generated using option "Flow". In this case the necessary flow information for the calculation of the mass transport is read from the intermediate flow result file. This option

can be useful if different quality calculations have to be performed all using the same flow calculations. This is the case if one or more parameters of the water quality model are changed during the calibration of the model, or if another process description file is used to generate another water quality model.

"B=Box" Using this option only the processes as defined in the process description file will be simulated. Transport is not calculated. The calculations will be performed for all defined sections within the network. Boundary conditions (both Flow and Quality) will not be used. If in the process description file flow variables are used, the default values from the *.mod file will be used. External variables (time dependent and location specific) will be used as defined in the quality menu "external variables". This option is useful when developing and testing a process description file. Furthermore the use of this option enables the examination of the relative importance of the transport processes in comparison with the other processes involved.

The result of the computation is stored in the defined result files (.res and .rek).

The calculation can be halted by pressing Ctrl S and aborted by pressing Ctrl C.

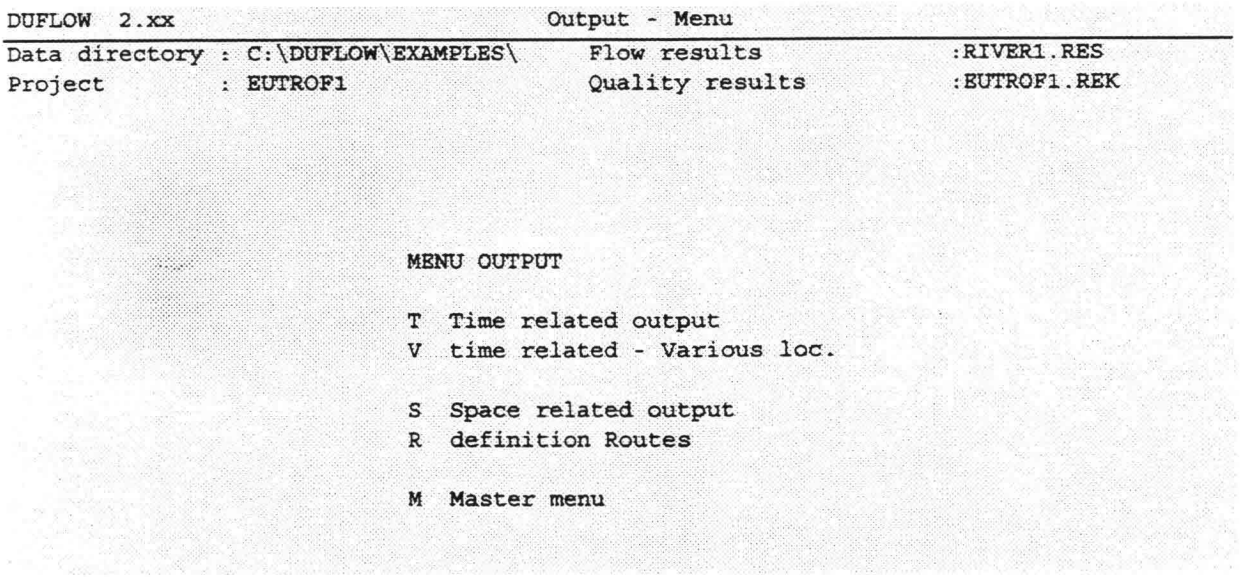
At the bottom of the screen the progress of the calculation(s) is displayed. Appendix F contains a list of errors.

3.6 Menu OUTPUT

Entered by selecting "output" from the "master menu".

Enables the presentation of the results in graphs or tables, as function of time or space.

The menu is shown below:



"T=Time related output"

Displays the results at specified location as a function of time (see paragraph 3.6.1).

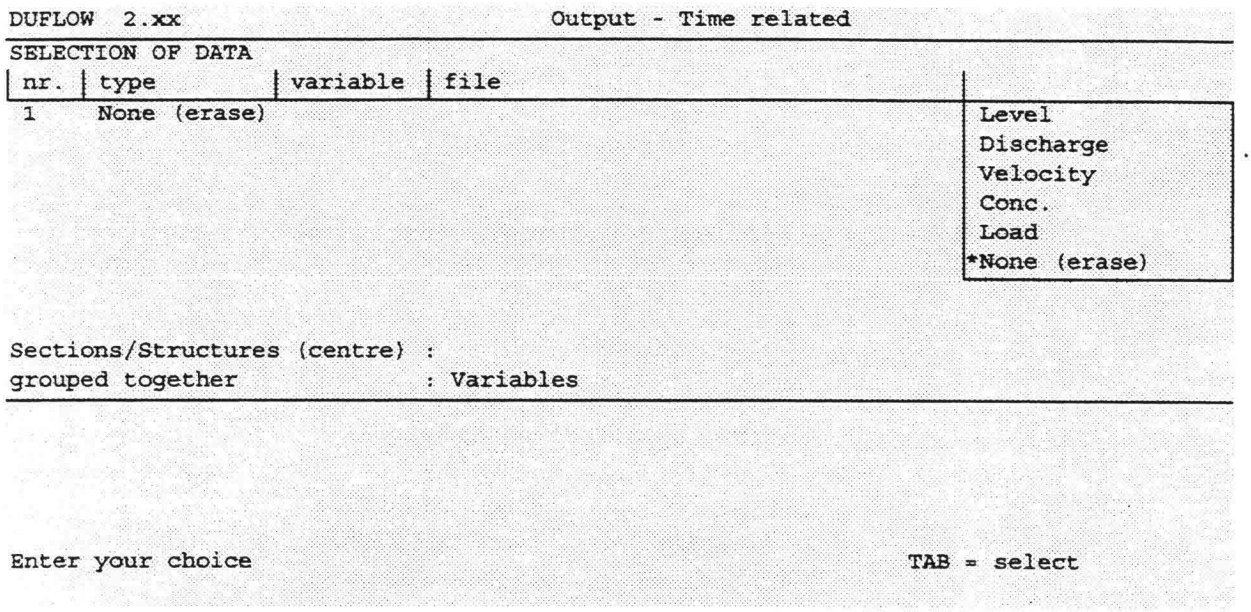
"V=time related - Various loc."
Enables graphs of several variables at various locations (see paragraph 3.6.2).

"S=Space related output"
Displays the results at specified time intervals along a defined longitudinal profile (route) (see paragraph 3.6.3).

"R=definition Routes"
To display space related output one or more routes or trajects through the network must be defined (see paragraph 3.6.4).

3.6.1 TIME RELATED OUTPUT

Entering the "Time related output", enables the presentation of the output as function time. The appearing table is shown below:



"type" From the pick list the type of variables to be displayed can be selected. Level, Discharge and Velocity always can be selected. The other available types are depending on the selection made in "QUALITY VARIABLES FOR OUTPUT" (see paragraph 3.4.1.3). the other types are:

"Concentration"	All state variables.
"Load"	Q•C for all state variables
"Function"	All intermediate results of function identifiers as defined in the *.mod file.

"variable" If the variable is pre-defined the program will enter the name of the variable, otherwise the user can select the desired variable from the pick list (see paragraph 3.2.4.1 for commands).

"file" The program will enter the name of the result file made during the project calculations. Any other result file or file containing

measured data can be entered. If they are stored in another data directory or drive then specify the full path. The result files in the project have the following extensions:
.res results flow model
.rek results quality model

"Sections/Structures"
The section numbers and structure codes (i.e. S1) must be entered. The selected variables will be displayed in the centre of the sections or structures. The available sections or structures are the ones selected in "locations for output". Type B after the section indication number to obtain the beginning of the section and E for the end.

When for the variable only "Levels" are selected the results will be displayed at the nodes, if "Levels" is selected in any other combination, the "Levels" will also be displayed at the middle of the sections.

"grouped together"

"Variables"	Make for each location a graph or a table with all variables.
"Locations"	Make for each variable a graph or a table with all locations.

After selection of the data, output can be directed to a table or a graph (see paragraph 3.6.1.1).

3.6.1.1 PRESENTATION OF OUTPUT

T=Table The data be will displayed in numerical way and can be written to a printer or file. Selection items are:

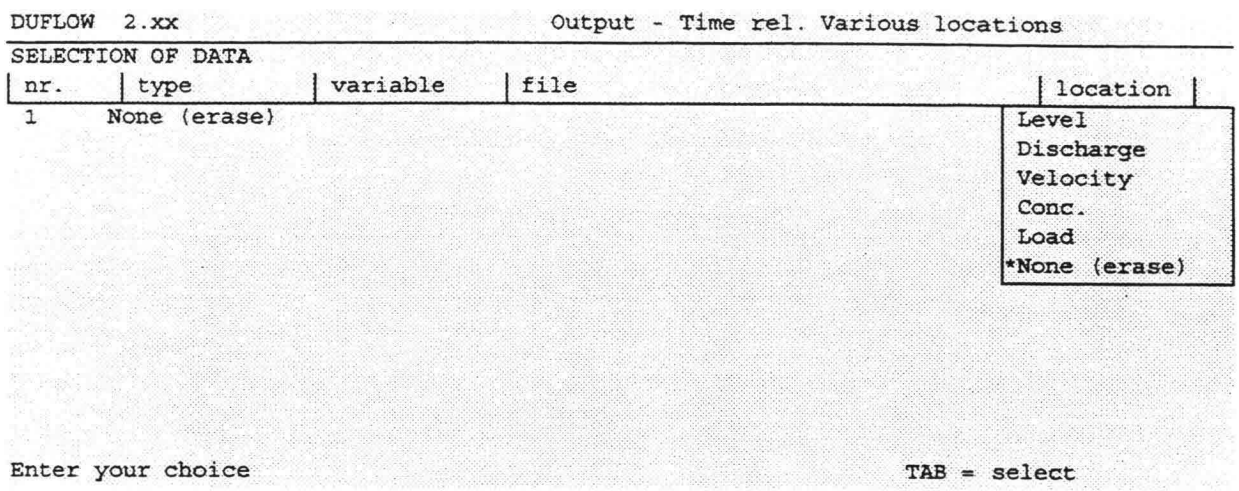
"Home, End"	Navigate in the table.
"PgUp, PgDn"	Navigate in the table.
←, ↑, ↓, →	Navigate in the table.
"Print"	The data can be printed to file or printer (see paragraph 3.2.4.4).
"PgUp"	Show the previous page on screen.
"PgDn,↵"	Show the next page on the screen.
"Esc"	Return to "time related output".

"G=Graph" The data will be displayed in a graph. Selection items are:
"File"
The graph will be written to a hpgl plot file. Enter the name of the plot file without extension.
"Plot"
The graph will be plotted on the connected plotter device.
"↵,S"
The graph will be displayed on screen.
"↑"
Edit the "layout of the graph". The default settings for minimum and maximum values and grid distance can be modified.

"Axes"
This option is only available, when the variables are of the same kind. In that case it is possible to make a choice on which side the Y-axis has to displayed.
If two different variables are selected the option will not be available, because both sides of the graph are needed.
Default is left.
"Esc"
Return to "TIME RELATED".

3.6.2 TIME RELATED - VARIOUS LOC.

Entering the "time related - Various loc.", enables the presentation of the output as a function of time. The appearing table is shown below:



"type" From the pick list the type of variables to be displayed can be selected. Level, Discharge and Velocity always can be selected. The other available types are depending on the selection made in "quality variables for output" (see paragraph 3.4.1.3). the other types are:

"Concentration"	All state variables.
"Load"	Q.C for all state variables
"Function"	All intermediate results of function identifiers as defined in the *.mod file.

"variable" If the variable is pre-defined the program will enter the name of the variable, otherwise the user can select the desired variable from the pick list (see paragraph 3.2.4.1 for commands).

"file" The programs will enter the name of the result file made during the project calculations. Any other result file or file containing measured data can be entered. If they are stored in another data directory or drive then specify the full path. The result files in the project have the following extensions:
.res results flow model
.rek results quality model

"Location" The section numbers, nodes and structure codes (i.e. S1) must be entered. The selected variables will be displayed in the middle of

the sections or structures. The available sections or structures are the ones selected in "locations for output". Type B after the section indication number to obtain the beginning of the section and E for the end.
When for the variable only "Levels" are selected the results will be displayed at the nodes, if "Levels" is selected in any other combination, the "Levels" will also be displayed at the middle of the sections.

After selection of the data, output can be directed to a table or a graph (see paragraph 3.6.1.1).

3.6.3 SPACE RELATED OUTPUT

Before this option can be used the output routes must be defined using "definition routes" (see paragraph 3.6.4).

"space related output" enables the presentation of output as function of space. The appearing table shown below:

```
DUFLOW  2.xx                                 Output - Space related
SELECTION OF DATA
| nr.   | type        | variable     | route     | file                    |
  1       None(erase)                                                     | Level
                                                                          | Discharge
                                                                          | Velocity
                                                                          | Conc.
                                                                          | Load
                                                                          |*None (erase)

first output                  : yymmdd    hhmm
last output                   : yymmdd    hhmm
interval                      :     dd    hhmm                # timepoints
grouped together              : Variables

Enter your choice                                                  TAB = select
```

"type" From the pick list the type of variables to be displayed can be selected. Level, Discharge and Velocity always can be selected. The other available types are depending on the selection made in "quality variables for output" (see paragraph 3.4.1.3). the other types are:

"Concentration"	All state variables.
"Load"	Q•C for all state variables
"Function"	All intermediate results of function identifiers as defined in the *.mod file.

"variable" If the variable is pre-defined the program will enter the name of the variable, otherwise the user can select the desired variable from the pick list (see paragraph 3.2.4.1 for commands).

"route" One of the defined routes can be selected. The default route is the first route defined in "definition routes" (see paragraph 3.6.4).

"file" The programs will enter the name of result file made during the project calculation. Any other result file or file containing measured data can be entered. If they are stored in another data directory or drive then specify the full path. The result files in the project have the following extensions:

.res results of flow model
.rek results of quality model

"first output" Starting time of the output. Default is "Starting of output" defined in "calculation definition" (see paragraph 3.4.1.1).

"last output" Ending time of the output. Default is "End of computation" defined in "calculation definition" (see paragraph 3.4.1.1).

"interval" Time step defined in days (dd), hours (hh) and minutes (mm). Default: "Output interval" (see paragraph 3.4.1.1)

"grouped together"

"Variables" Make for each location a graph or a table with all variables.

"Locations" Make for each variable a graph or a table with all locations.

After selection of the data, output can be directed to a table or a graph (see paragraph 3.6.1.1).

3.6.4 DEFINITION ROUTES

To display space related output one or more routes or trajects through the network must be defined. A route is not necessarily continuous.

The maximum number of routes that can be defined is 10.

Entering the "definition routes" the following table will appear;

```
DUFLOW  2.xx                    Output - Definition routes
Route      nr:
Route name  :
| Section |          | X-begin | X-end |                    | node beg. | node end |
```

"Route name" For identification of a route a text can be entered. The identification will be shown in the table of "space related output".

"Section" To plot space related output the route(s) are defined by specifying the sections (or structures), available are only those selected in "locations for output" (see paragraph 3.4.1.2). If the section number is negative the section is placed in the route back to front (The two most right most columns show the connecting nodes). Terminate with entering 0 as section number.

"←,↑" Edit mode enables to modify the already existing route.

"X" Enables to modify the given X co-ordinates.

"↵" Go to the next route number.

"Esc" Return to "menu output".

3.7 Menu FILE NAMES SPECIFICATION

This option enables the definition of projects. A project links all input file names necessary to perform a simulation. The input table is shown below:

```
DUFLOW  2.xx                          File names specification


        Data directory      : C:\DUFLOW\EXAMPLES\
        Project                       : EUTROF1     (.PRJ)

        Control data                  : RIEUT1      (.CTR)
        FLOW
          Network data                : RIVER1      (.NET.NOD)
          Initial conditions          : RIVER1      (.BEG)
          Boundary conditions         : RIVER1      (.BND)
          Results                     : RIVER1      (.RES)
        QUALITY
          model                       : EUTROF1     (.MOB)
          Initial conditions          : RIEUT1      (.BEK)
          Boundary conditions         : RIEUT1      (.BNK)
          External variables          : RIEUT1      (.EXT)
          Parameters                  : RIEUT1      (.PRM)
          Results                     : EUTROF1     (.REK)


Enter new path for data files
```

To start DUFLOW for the first time, the required files to store the information can be entered. The "Data directory" should have been already defined.

Specify filenames without extension. The extensions will be added by the program:

.PRJ	for the project file
.CTR	for the control data file
.RTS	for the route file
.NOD .NET	for the network files
.BEG	for the hydraulic initial conditions file
.BND	for the hydraulic boundary file
.RES	for the hydraulic result file
.MOB	for the model file
.BEK	for the quality initial conditions file
.BNK	for the quality boundary file
.EXT	for the external variables file
.PRM	for the parameter file
.REK	for the quality result file

If one or more existing projects are available on the "Data directory", the name of project file can be entered. The program will prompt:

"Read new project file? (Y/N)"

Yes All required files for the project will be loaded.

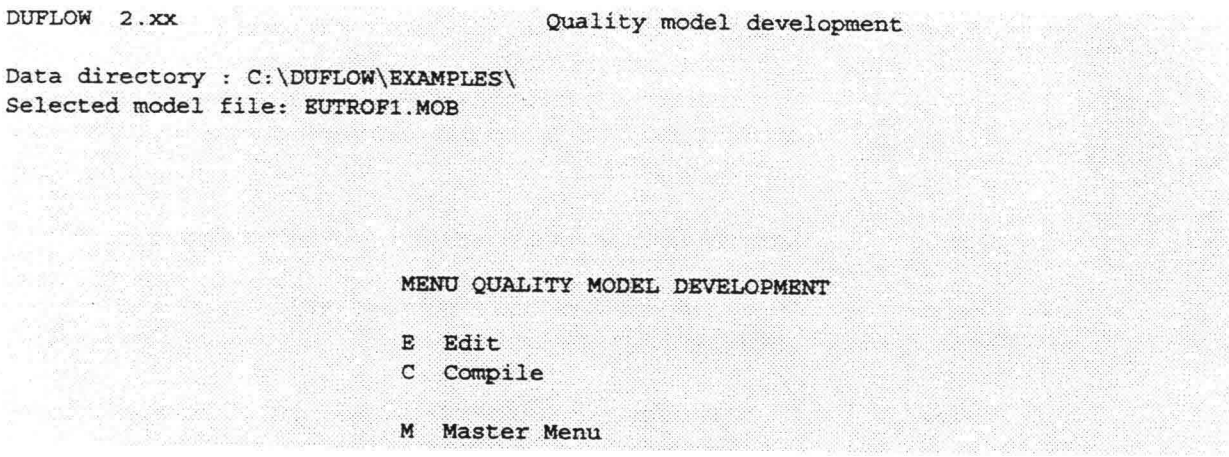
No The files will not be loaded and the files have to be entered by the user.

After entering the last file name or pressing Esc the program will return to "master menu"

3.8 Menu QUALITY MODEL DEVELOPMENT

The interactions involved without transport processes, advection and dispersion, have to be supplied in this part of the program. These are stored in the process description file *.mod. The resulting set of equations has to be compiled using DUPROL. After compilation a *.mob file is created which is read by DUFLOW.

DUPROL is not case sensitive.



"E=Edit" Enables the user to create new or edit an existing file containing process description (*.mod). DUFLOW does not supply internal editing facilities. Editing is performed using an external editor (see paragraph 3.8.1).

"C=Compile" Compiles the process descriptions and generates *.mob file, which is read by DUFLOW.

3.8.1 EDIT

3.8.1.1 SYNTAX

Some common rules of syntax:

An *identifier* (name of a variable) starts with a character and consists of a maximum of six characters. Non alpha-numerical characters are not allowed. The name of variable *d* is reserved for the dispersion coefficient and should not be used.

Comments start with /* and end with */. Comments are only allowed at the beginning of the file.

A model file consists of two parts:

- Declaration section
  In this part the different variables are defined (see paragraph 3.8.1.2).
- Compound statement
  This part contains the equations describing the processes (see paragraph 3.8.1.3).

The *.MOD must be closed with an empty line at the bottom, otherwise DUPROL cannot compile the file correctly.

3.8.1.2 DECLARATION SECTION

In the declaration section all different variables have to be defined. Five types of variables are distinguished:

water Water column state variables. The flow has affect on this type of variable.

bottom Sediment state variables. For these type of variables horizontal transport is omitted. Hence only the processes are calculated. Exchange between the sediment and the overlying water column should be described by the user. The flow has no effect on this type of variables.

xt External variables, which are space and/or time dependent.

parm Parameters, constants and coefficients used in the process equations.

flow Flow variables, supplied by the hydraulic part of the model. These variables differ from the other variables by the fact that the identifiers of these variables are built-in. The following identifiers are available:

Z	Depth of water.
Q	Flow.
U	Water velocity.
As	Flow area.
Ab	Storage area.
Bs	Flow width.
Bb	Storage width.
dx	Half of the length of section.
V	Half of the volume of section.
Wf	Wind velocity.
Wd	Wind direction.

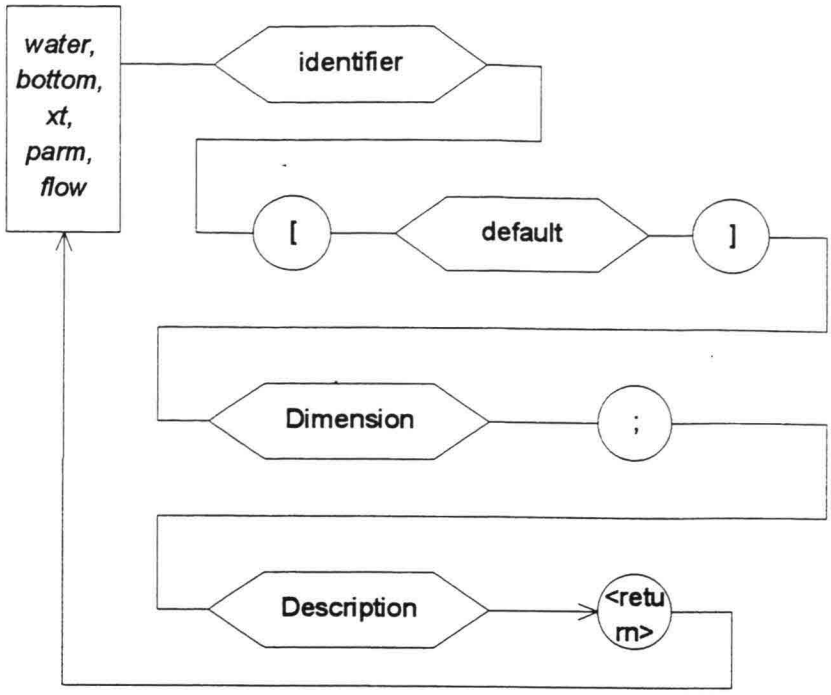


Figure 3.7 Syntax of the declaration section.

Default — Default values are used if the user does not supply values within the other menus. For the state variables the default values are used if the user does not enter initial conditions. The default value has to be given in between brackets '[' and ']'.

Dimension — The unit of the variable may consist of a maximum of 10 characters, followed by ';'. Longer names will be truncated.

Description — The description of the variable may not exceed a maximum of 40 characters. Longer lines will be truncated.

Below a part of the declaration section is given of the model description file.

```
/* Simple declaration section for explaining DUFLOW 2.00      */
/* remarks: No physical meaning        !!!                    */
/* T.H.E. Editor, December 1995                               */

water   A       [ 2.000]  ug-C/l          ;Algal biomass
water   O2      [10.000]  mg/l            ;Oxygen
water   BOD     [ 5.000]  mg-O2/l         ;BOD
water   SS      [10.000]  mg/l            ;Suspended solids

parm    kp      [ 0.005]  mg-P/l          ;Monod-constant P algal growth
parm    kn      [ 0.010]  mg-N/l          ;Monod-constant N algal growth
parm    ealg    [ 0.016]  ug-Chl/l,m      ;Specific extinction algae
parm    e0      [ 1.000]  1/m       ;Background extinction of the water

xt      i0      [ 10.00]  W/m2            ;Irradiation
xt      t       [ 20.00]  oC              ;Temperature
xt      resf    [  0.50]  g/m2.dag        ;Resuspension  flux
```

3.8.1.3 COMPOUND STATEMENT

3.8.1.3.1 INTRODUCTION

In this part the process descriptions have to be included. This section starts with '{' and should be closed '}'. All arithmetic expressions may be used (see next paragraph). The way the differential equations for the state variables should be entered needs some additional explanation.

For most state variables the kinetic derivative has the following form:

$$\frac{\partial C}{\partial t} = k_1 C + k_0$$

In this equation all first and zero order terms should be separated. For example the following equation:

$$\frac{\partial C}{\partial t} = k_a(C_s - C) - k_d L$$ should be rearranged like:

$$\frac{\partial C}{\partial t} = -k_a C + k_a C_s - k_d L$$

Internally lumped first and zero order coefficient are used, which should be defined by the user as:

$$k_1(C) = -k_a$$

$$k_0(C) = +k_a C_s - k_d L$$

If the k_1 and k_0 coefficient are not defined they will be set equal to zero.

For non state variables a function identifier is used. The declaration of these type of variables is implicit, which means that they may not be declared in the declaration section.

Three types of compound statements are available:

- formula See paragraph 3.8.1.3.2
- if-statement See paragraph 3.8.1.3.3
- iteration-statement See paragraph 3.8.1.3.4

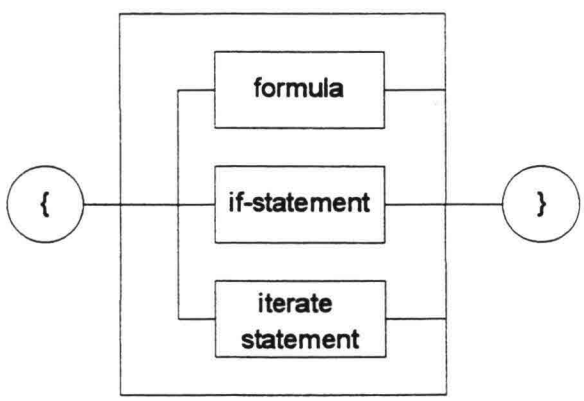


Figure 3.8 Syntax of the compound statement.

3.8.1.3.2 FORMULA

The general syntax of a formula is shown at figure 3.9.

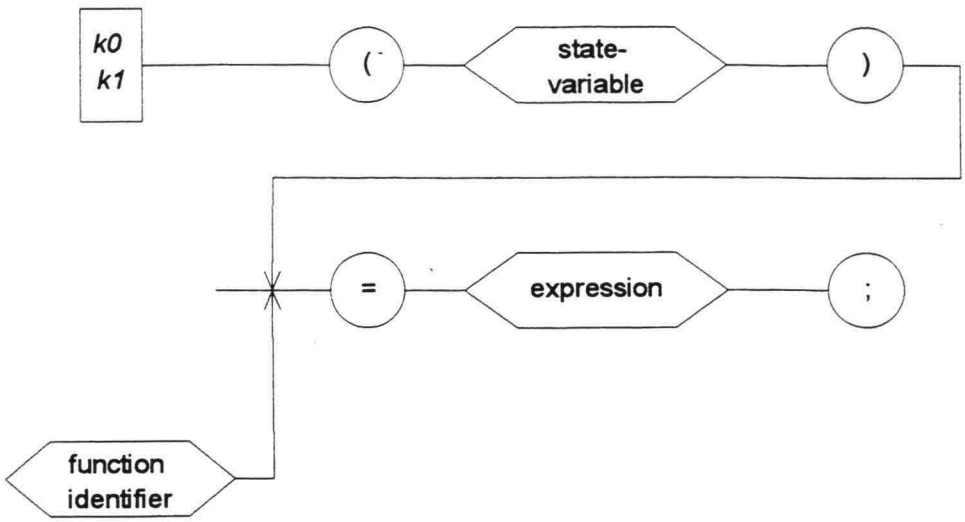


Figure 3.9 Syntax of Formula

k1	First order reaction coefficient.
k0	Zero order reaction coefficient.
state variable	water or bottom state variable.
function identifier	Identifier of a function.
expression	Definition of a function. A formula, defining the function identifier at right side of the equation. This formula consists off regular mathematic operators completed with several built-in functions . A function identifier is not allowed to appear in the right side of his own definition. A function identifier must already have been defined before it can be used in the right side of an equation. The following operators are available (in order of priority):

Arithmetic :	Built -in functions:
()	sin(x)
^ (to invoke)	cos(x)
*	tan (x)
/	exp (x) (ex)
+	ln(x)
-	log (x) (^{10}log(x))
	abs (x) (| x |)
	min(x1,x 2,..., xn)
	max (x1, x2,... ,xn)

3.8.1.3.3 IF-STATEMENT

DUPROL contains a flow-control statement. The syntax of this statement is shown in figure 3.10.

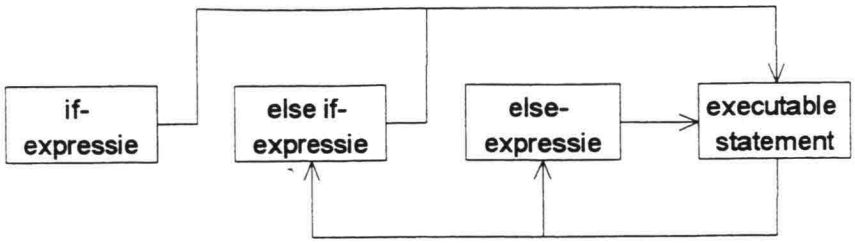


Figure 3.10 Syntax if statement

The *if statement* can be applied in several ways. The most common use of the statement will be shown in the following examples.

Example 1

Most basic formulation. If the condition in '(and is true the statement between the '{' and '}' will be executed.

```
if (NO3==0.0)
{
      pnh4=0.0;
}
```

Example 2

Example 1 can be extended with an alternative executable statement if the condition is false.

```
if (NO3==0.0)
{
      pnh4=0.0;
}
else
{
      pnh4= NO3/NH4*Kmn;
}
```

Example 3

If statements can be applied in several complicated situations, even nesting is allowed.

```
if (NO3==0.0)
{
      pnh4=0.0;
}
else if (NH4==0.0)
{
      pnh4= NH4/NO3*Kmn;
      if (Kop>=1.0) AND (Lap!=0.0)
      {
            Mip=pnh4*Kmn/2+0.001;
      }
}
else
{
      pnh4=NO3/NH4*Kmn;
}
```

General remarks concerning the use of if statements :

- Conditions can be defined using the following relational and logical (in order of priority):
  >
  >=
  <
  <=
  == (is equal to)
  != (unequal to)
  ! (not)
  AND
  OR
- Also in an if statement every executable statement must be closed with a ';'.
- Between the '{ }' the user can define a block of executable statements (= compound statement).

3.8.1.3.4 ITERATION STATEMENT

In figure 3.11 a syntax of an iteration statement is shown.

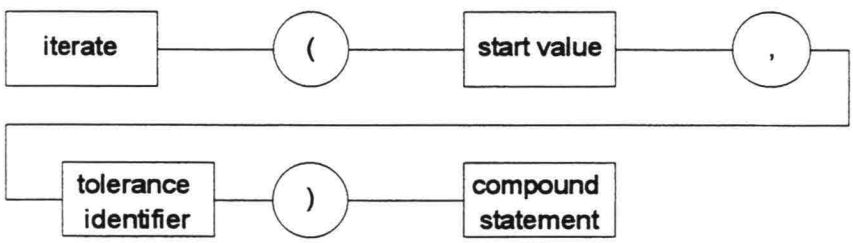


Figure 3.11 Syntax of iteration statement

The statement exists of:

start value	The start value of the variable of the iteration.
tolerance identifier	This variable (parameter) contains the maximum allowed difference between the two succeeding iteration results.
compound statement	The formula which has to be approximated by means of iteration.

An example is the iteration of the wavelength at depth D:

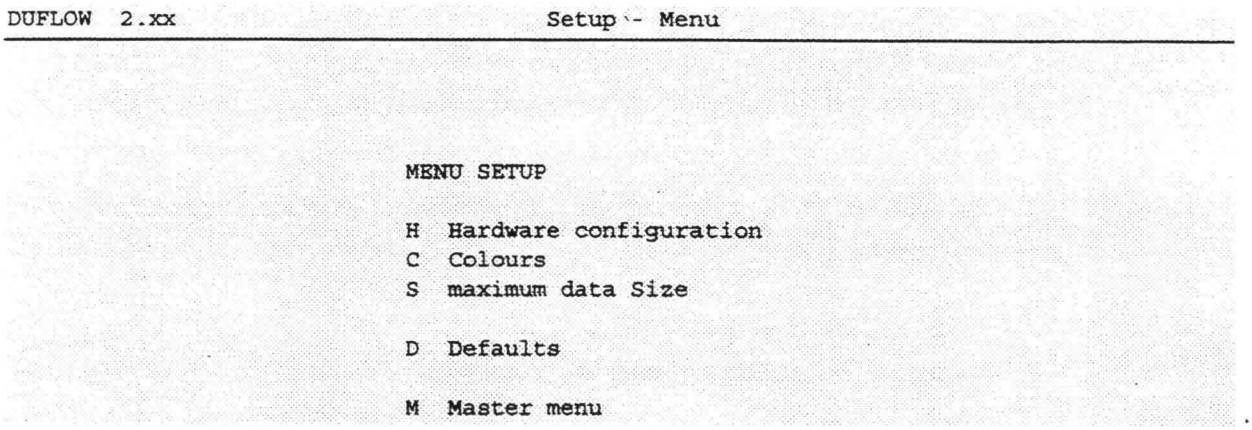
```
iterate       (LD=1.0, eps)
{
      LD= L * ((exp(+2*pie*D/LD))-(exp(-2*pie*D/LD) ))/
              ((exp(+2*pie*D/LD))+(exp(-2*pie*D/LD) ))
}
```

In which:

LD=1.0 , start value of the variable;
eps , tolerance identifier;
LD= Function (L_D)
, function to be iterated;
pie , $\pi = 3.1415927$;
L , wavelength at the surface;
D , depth.

3.9 Menu SETUP

The "SETUP" is shown below:



"Hardware configuration"
Adapt DUFLOW to your configuration (see paragraph 3.9.1).

"Colours" Set the colours used by the program (see paragraph 3.9.2).

"maximum data Size"
Specify maximum amounts of data to be handled by Input and Output parts (see paragraph 3.9.3).

"Defaults" Overwrite the setup of the three items with defaults. In each of the three items there are "default" options for that part only.

"Master menu"
Return to the "master menu". If any change has been made the user is prompted for saving the changes in the configuration file.

3.9.1 HARDWARE CONFIGURATION

To install the program see appendix A, for further explanation about the hardware configuration consult the guides of the installed devices.

"Printer present"
If Yes is answered the program gives an option to print the output to the connected printer device. If no printer is connected the output will be send to a file on disk.

"Printer port (LPTn)"
Defines the port name for the printer connection.

"Plotter present"

If Yes is answered the programs will direct all plot commands to the connected plotter device. The plotter must be able to handle hpgl files.

"Port (COMn,baud,par,data,stop)"

If data is send immediately to the plotter the setting of the serial communication must be given: port name, baud rate, parity(o/e/n), number of data bits, number of stop bits (e.g.: com1,9600,E,7,1). These settings must correspond to the dipswitch settings of the plotter.

"Graphical card"

Choose the right graphical card from pick list.

"# lines on printer page"

When performing output to the printer the program gives a Form Feed command after the specified number of lines.

"Path editor" Enter the name of the editor which will be activated in "quality model development". (The editor program must be installed).

3.9.2 COLOURS

"Text colours"

Choose a number from the sample table for each of the three text types. If 128 is added to the number the text will be blinking.

"Screen graphics"

Depending on the selected graphical mode a number of colours can be selected for the graphical output to the screen. The intensity of the samples may differ from the real colour in the graphs.

"Plotter graphics"

For output to the plotter (or plotter file) the numbers of the pens to apply must be given.

3.9.3 MAXIMUM DATA SIZE

```
DUFLOW  2.xx                          Setup - Maximum Data Size

MAXIMUM DATA SIZE


Maximum size of input data (#values)    : 10000
Maximum number of boundary conditions   :    50
Maximum number of lines to plot         :    30
Maximum number of output steps          :  1000
```

"Maximum number of input data (# values)"

The amount of memory that the input part of the program allocates to store the data. If the value is too small than DUFLOW

may generate a `Memory Full' error or a (fatal) `Subscript out of range' error. If it is too large an error message `Out of memory' or `Out of string space' will occur after choosing "Input" from the "master menu". The maximum value is approximately: (max. memory size - 330000)/4.

"Maximum number of boundary conditions"

The maximum number of series that the input part must be able to handle for each of the items flow boundary conditions, structure operations, quality boundary conditions, external variables and measured data. This value is not critical; it may be far too large.

"Maximum number of lines to plot"

"Maximum number of output steps"

The amount of memory that the output part of the program allocates to store the selected result data.

In "Time related output" the number of locations is restricted if the number of selected variables times the number of selected locations is more than (3). The number of timepoints is truncated if it exceeds (4).

In "Space related output" the number of time levels is truncated if (3) times (4) is larger than the number of selected timepoints times the largest number of sections in the selected routes times 2. The maximum of (3) times (4) is approximate: (maximum memory size - 200000)/8.

4. Simplified examples

4.1 Introduction

The duflow package has been subjected to two series of tests. The first series contains academic tests in which one specific effect is studied in each test; these tests are not reported here. The second series contains simplified actual cases which are representative for the type of problems that can be handled by duflow.

Three selected problems are presented:

- Tidal motion in the Eastern Scheldt
- Drainage system in Bangladesh
- Flood wave propagation at the river Meuse

Each subsequent part of this chapter treats one application and includes the general characteristics, schematization, choice of boundary conditions and results of computation. Optionally the sensitivity with respect to variations of certain physical effects is discussed.

Most of the graphs are generated by duflow and give a clear impression of the output options of the package.

The data of the discussed examples are available on a separate data diskette which allows the user to verify the results and compute the consequences of certain variations.

4.2 Tidal motion: Oosterschelde

Problem characteristics

The Oosterschelde or Eastern Scheldt is an estuary located in the Southwestern part of the Netherlands. This estuary is connected with the North Sea through a storm surge barrier which was completed in 1986. Shortly after the completion of this barrier, the eastern part of the closed estuary, which is part of the connection between the port of Antwerp and the river Rhine, was separated permanently from the main part of the estuary. Previously this part was connected to the main estuary by two openings known as the Krammer and the Tholense Gat. Closing the gates was done using sand which did not allow for flow velocities higher than 2 m/s during this operation. The purpose of the computations presented here is to investigate how sensitive the velocities in the openings are to manipulations with the gates of the storm surge barrier.

The Oosterschelde estuary is approximately 50 km long and 5 to 6 km wide. It has a number of deep gullies of 1 to 2 km wide, which are up to 30 or 40 m in depth, and wide shallow intertidal flats in between. The area is shown in figure 4.1 together with the schematization. In this figure S1 represents the storm surge barrier which has 62 openings of 45 m width each, which can be closed in case of storm surges. The maximum cross-sectional flow area of these openings is approximately 24000 m2 at mean sea level. In the case presented here, the opening is reduced to 16000 m2 by closing some of the gates in order to diminish the velocities in the Krammer (S3) and Tholense Gat (S2).

The schematization used in this example is a simplified version of a more detailed schematization which contains 250 branches and was calibrated accurately. The present schematization has 11 open channel sections and 3 structures (S1, S2 and S3). The tidal seaside boundary value was taken from a gauge located in the estuary entrance (near node 1 in figure 4.1). The largest difference between the simplified and the extended model in terms of computed water level was 10 cm, usually the difference was not more then 2 or 3 cm. Thus the simple model was considered adequate for a sensitivity analysis.

Analysis

Two situations were considered:

- case a where both the Krammer and the Tholense Gat were open;
- case b where the Tholense Gat was closed and the Krammer was open.

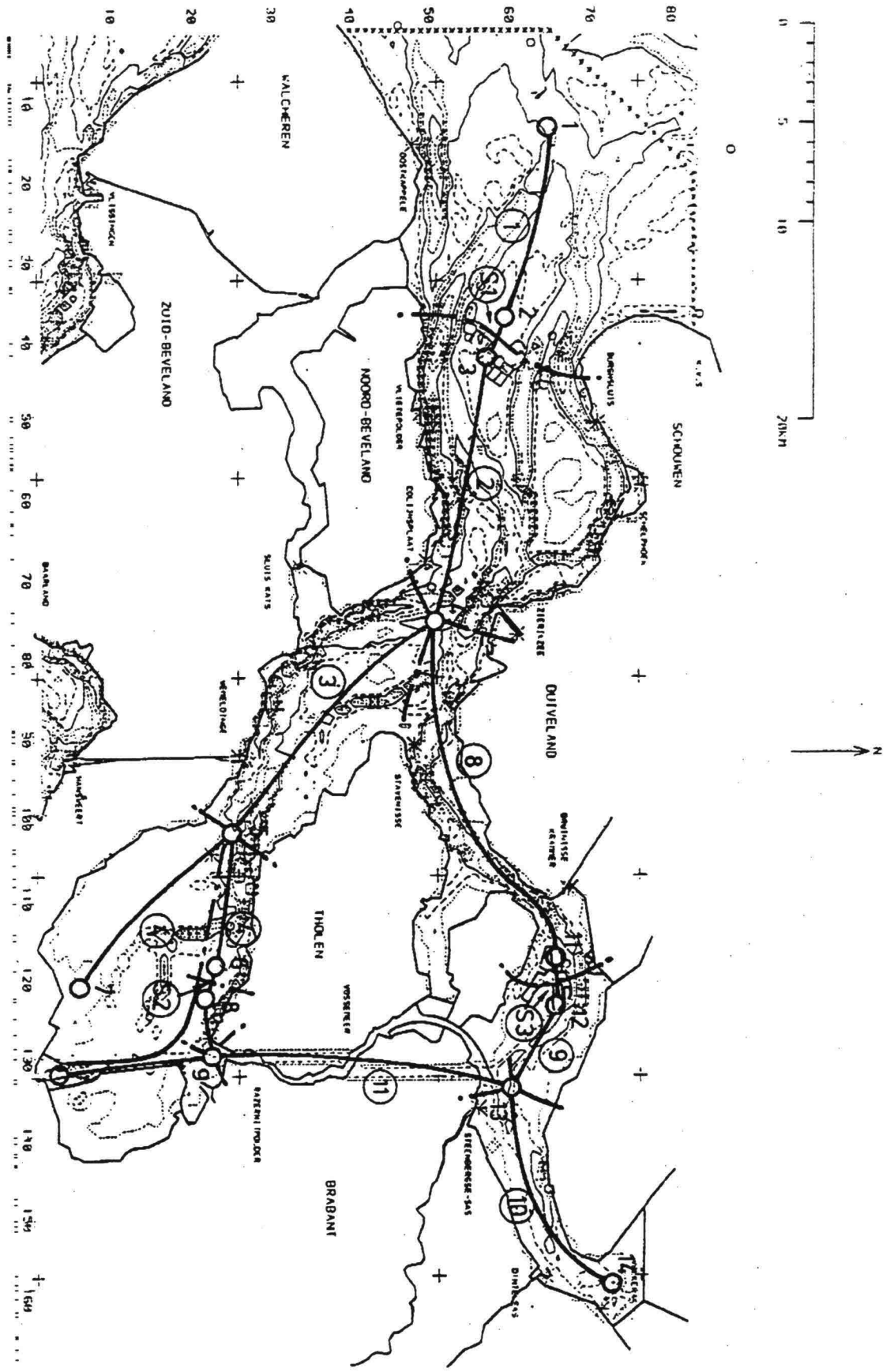


Figure 4.1 Schematization of the Oosterschelde estuary. Wide lines represent river sections, dashed lines boundaries between sections. S1, S2 and S3 are structures

Figure 4.2 shows the water levels in nodes 1 (seaside boundary), 3 (immediately behind the storm surge barrier), 13 (in the Krammer) and 9 (in the Zoommeer), both for case a and b.

Figure 4.3 shows the discharge through openings S2 and S3; obviously in case b the discharge through S2 is identically zero. It is seen that in case b the difference in water level between nodes 9 and 13 is much larger than in case a, and that the discharge in S3 increases as a result of the closure of S2.

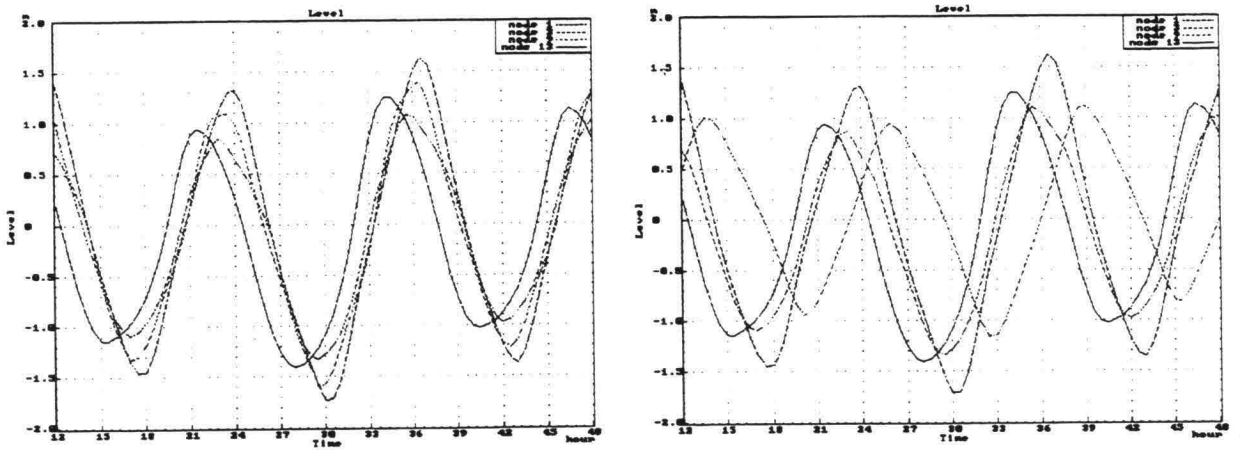


Figure 4.2 Water levels as function of time in nodes 1, 3, 9 and 13 (for thier locations see figure 4.1) Panel a: S2 and S3 open; Panel b: S2 closed

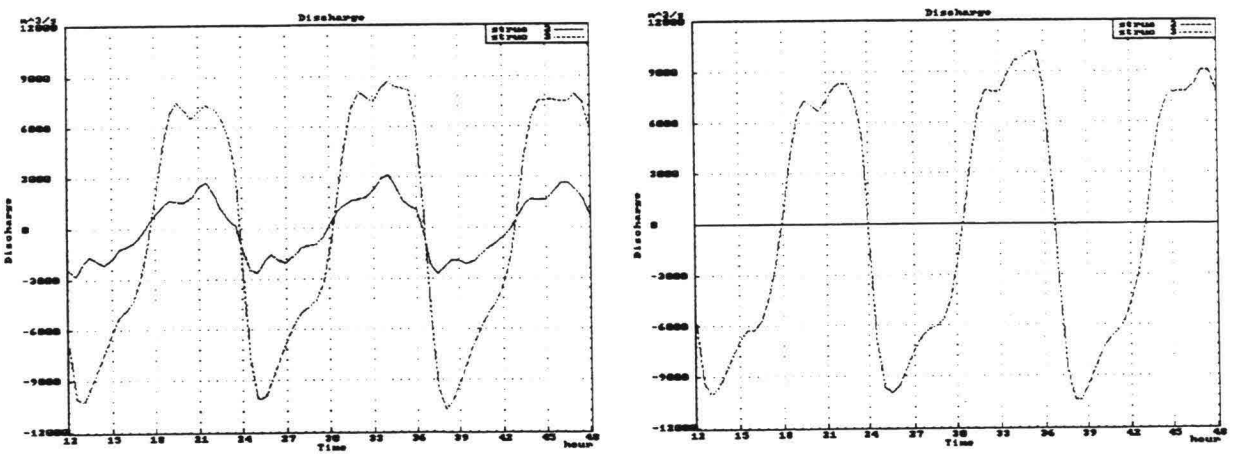


Figure 4.3 Discharges in openings S2 and S3 as function of time. Panel a: S2 and S3 open; Panel b: S2 closed

4.3 A drainage system

Problem characteristics

A polder in the Southwest of Bangladesh is used mainly for rice culture. The drainage system in the polder consists of a number of primary canals which are included in the schematization (see figure 4.4). Secondary canals are not modelled explicitly and only their storage capacity is taken into account. At the outer side the sluice discharges into a tidal river.

Purpose of the study is to optimize the design of the sluice (culvert type).

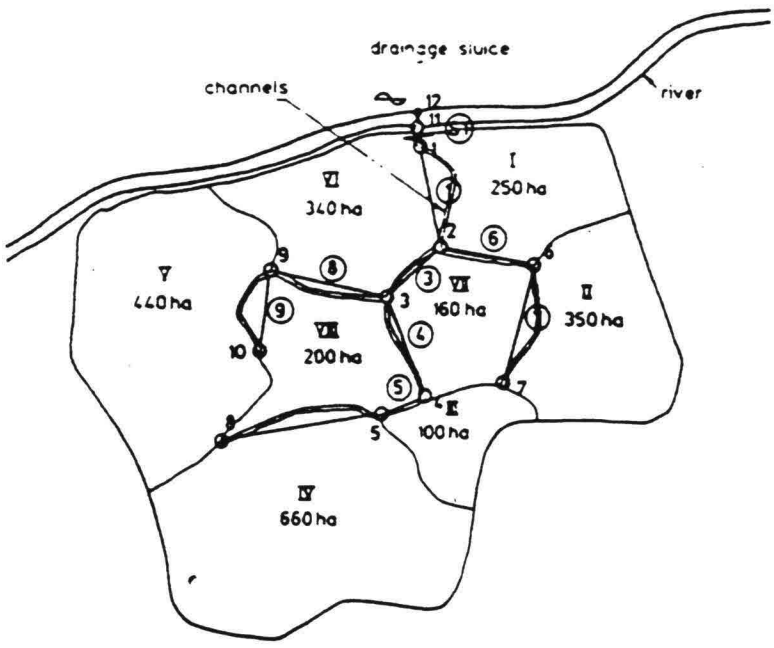


Figure 4.4 Map of the polder and schematization

First a compution was made for a rough estimate of the sluice dimensions. Next computations were carried out using duflow to optimize the dimensions by varying the invert level (sill level), width and top level of the culvert.

The total area of the polder is 2500 ha, divided in eight subregions with different surface levels. From these catchment areas the water discharges into the main drainage system which has a storage area of approximately 6 ha in the primary canals. The storage area of secondary canals and local depressions is assigned to the nearest primary canals (table 4.1).

Section	area (ha)	add.storage (m^2)	length (m)	add.with
1	215	29137	1600	18.21
2	85	11519	900	12.80
3	80	10842	1400	7.74
4	80	10842	600	18.07
5	710	96219	2300	41.83
6	180	24394	1200	20.33
7	450	60984	1400	43.56
8	220	29814	1400	21.30
9	480	65050	1100	59.14
	2500	338800		

Table 4.1 Area, additional storage, length and the additional width of the sections

The design drainage level in the canals is 1.00 m above project reference level, variations of a few decimeters are allowed. The maximum storage level allowed is 1.30 m above reference level, the design drain is 30 mm/day.

The catchment run-off (from design drain) is assumed to be concentrated at the nodes. The size of the catchment area corresponding to each node is shown in table 4.2.

Node	Catchment area (ha)
1	108
2	240
3	192
4	80
5	395
6	315
7	350
8	355
9	225
10	240
	2500

Table 4.2 Catchment areas

A first estimate of the sluice dimensions

The main parameters to be estimated are the sill level and the width. First the maximum discharge through the sluice is determined. The sluice is closed when the water level in the tidal river is above that in the drainage canals. When the sluice opens the polder water level will be at its maximum of +1.30 m and at the end of the opening period around the design level of +1.00m.

If the average water level in the polder is assumed to be around +1.15 m then the time of opening of the sluice is 7.3 hours (figure 4.5). During one tidal cycle of 12.5 hours an amount of 25*106*12.5*0.03/24 = 39000 m3 drains into the channel system with an average discharge of 14.8 m3/s. Experience shows that

the maximum discharge is appr. 15% higher than this average and equals 17.5 m3/s.

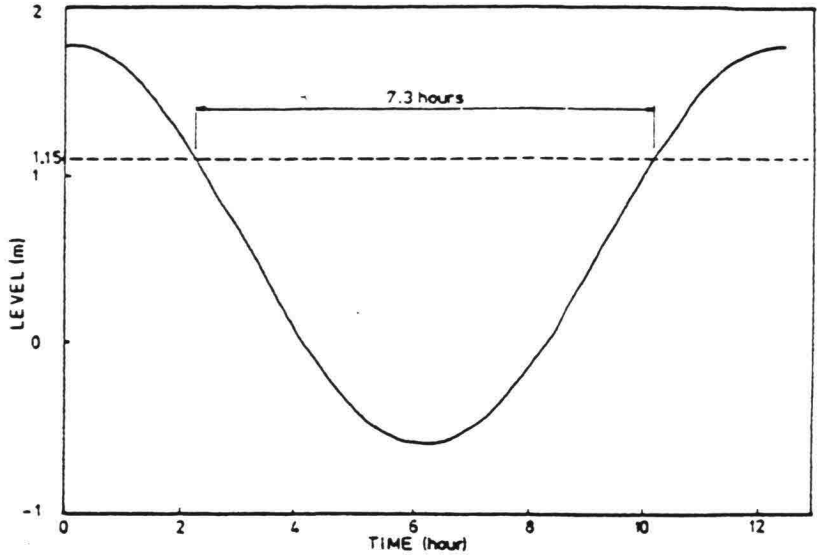


Figure 4.5 Opening time of sluice during one tidal period

To prevent damage to the (wooden) structure the maximum velocity in the sluice is 3.5 m/s. This maximum will be reached shortly after opening at the transition from sub-critical to critical flow since the level in the river drops rapidly.

Critical flow is described by:

$$v = \sqrt{\frac{2}{3} g (H - H_{sill})}$$

Substituting = 3.5 m/s results in (H - Hsill) = 1.87 m. Assuming the water level just upstream of the weir to be + 1.15 m, then the sill level is determined at -0.80 m. The corresponding width equals 5 m.

The top level of the culvert, (in the model represented as the gate level of a submerged weir) is the third parameter describing the sluice. This level is determined by increasing the water depth at entry above sill level by 20%, which implies that the water most of the time flows free of the top of the culvert.

Analysis

The duflow computations are done with a constant upstream discharge of 14.8 m3/s and a sinusoidally varying water level in the tidal river. Two results are shown. Figure 4.6 shows the water level at both sides of the sluice and the discharge through the sluice during a tidal period. Figure 4.7 shows the water level at one of the extremities of the network together with the water level upstream of the sluice. This shows that after opening of the sluice a difference in level develops between the extremities of the network and the sluice, which is reduced to zero after closure of the structure.

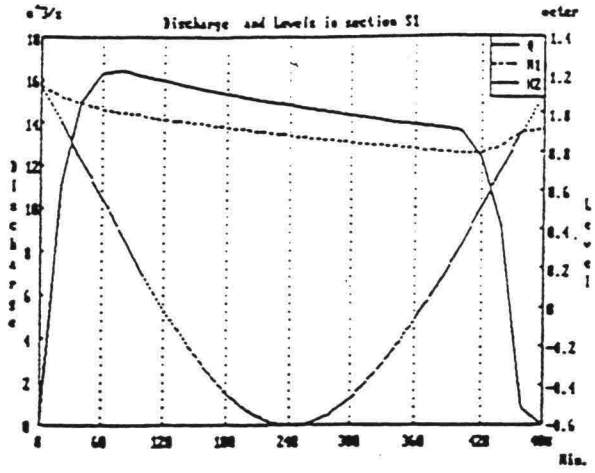


Figure 4.6 Levels and discharge during low tide

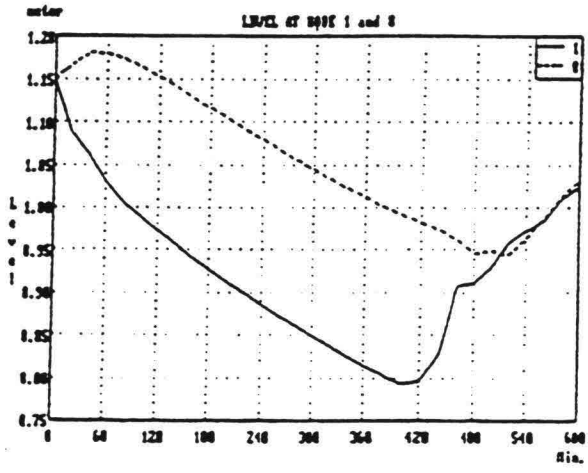


Figure 4.7 Levels at sluice and polder during one tidal period

4.4 Floodwave simulation: Belgian Meuse

Problem characteristics

The river Meuse (Maas) originates in France, flows through Belgium and the Netherlands, where it ends in the south-western delta area. Its total stretch is about 900 km with a catchment area of about 33500 km2 (fig 4.8).

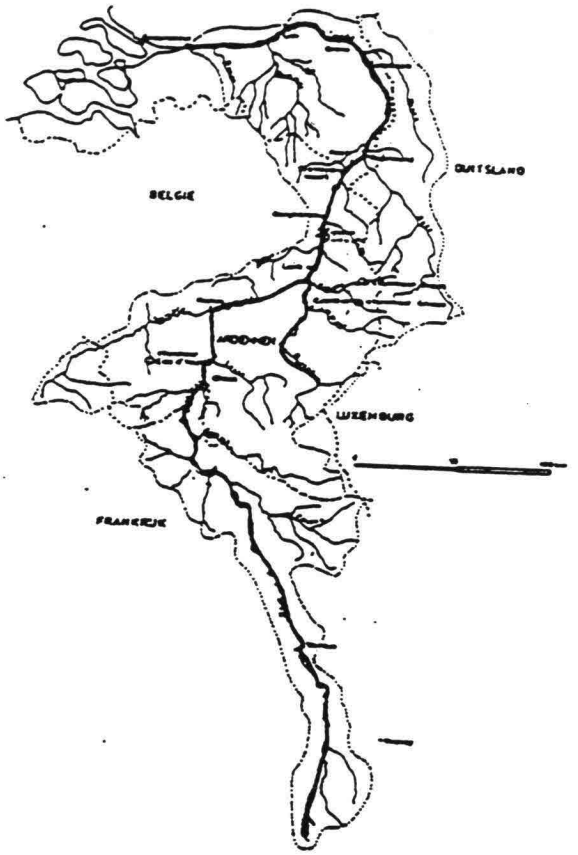


Figure 4.8 River Meuse

The river has a pluvial regime mainly fed by rain induced run-off in the upper parts of the catchment.

Especially in the Belgian part of the river major hydraulic works have been carried out to control water levels with respect to flooding, navigation and hydropower development.

The system described in this section includes a river stretch of some 60 km from upstream Liege (Belgium) to Borgharen (The Netherlands), with an average width of 110 m and depth of 8 m, various tributaries and diversions, six control structures some of which with movable gates and a number of bridges. Flood storage capacity in this river stretch is negligible.

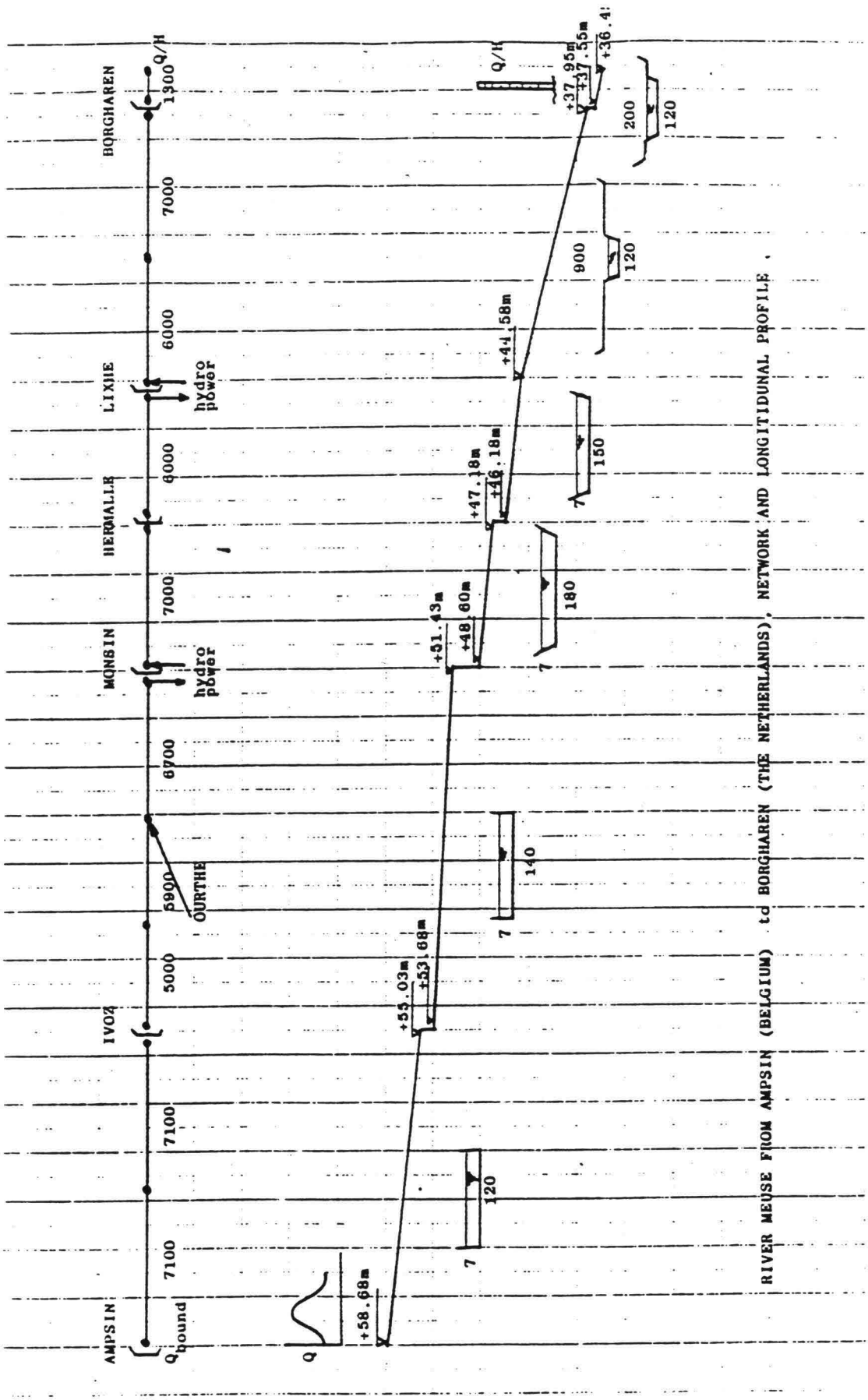
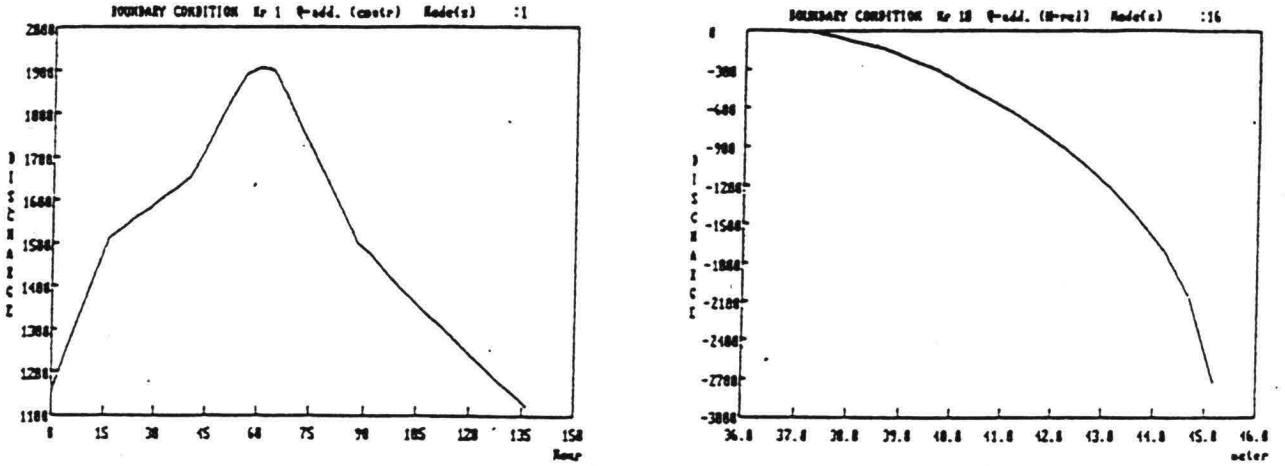


Figure 4.9 River Meuse from Ampsin (Belgium) to Borgharen (The Netherlands), network and longitudinal profile.

One of the objectives of the study was to determine operation rules for the existing weirs under flood conditions. Criteria were to be maintained water levels (navigation, hydropower), discharges (sediment and pollutant transport) and translation of the flood waves. Results of the computation may enable to develop optimal operation procedures for the structures under high and medium flow conditions.

Although rough indications for the operation of structures are given, their actual operation is unknown. This example intends to derive such an operating policy.

For the actual study a detailed model was constructed which contains fifty channel sections, six control structures, sixty-two nodes, two major tributaries and two diversions. An accuracy check was made using a simplified model. This simplified model is discussed here and consists of twelve sections, the major tributary Ourthe and two diversions for run-of-the-river hydropower generation (figure 4.9). The boundary condition (figure 4.10) at the upstream end is the 1984 flood hydrography (at Ampsin, time dependent discharge), and at the downstream end a Q-H relation (downstream of weir Borgharen, steady state rating curve). Cross-sections were defined as specified in figure 4.9, the De Chézy roughness coefficient was set to 45 $m^{1/2}/s$.



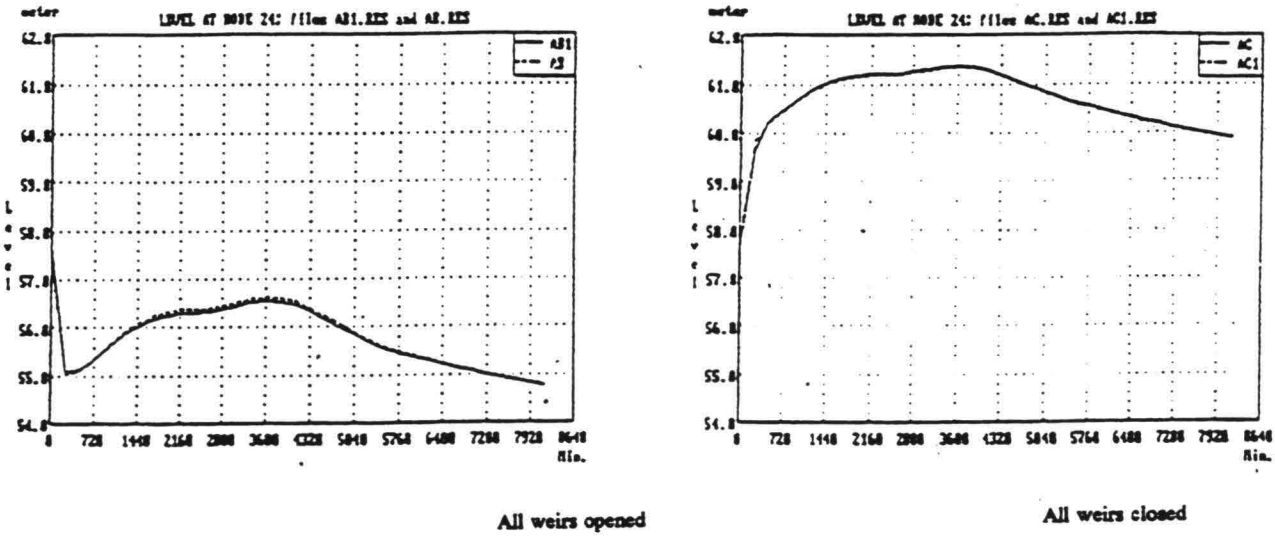
Upstream Downstream

Figure 4.10 Boundary conditions

Analysis

First test runs were made to compare the accuracy of the simplified model under the two extreme operating policies: all structures opened and all structures closed. The simulated water levels upstream the weir of Monsin are given in figure 4.11, the water levels at the downstream boundary condition in figure 4.12. It was concluded that for flood conditions the simplified model provided accurate results.

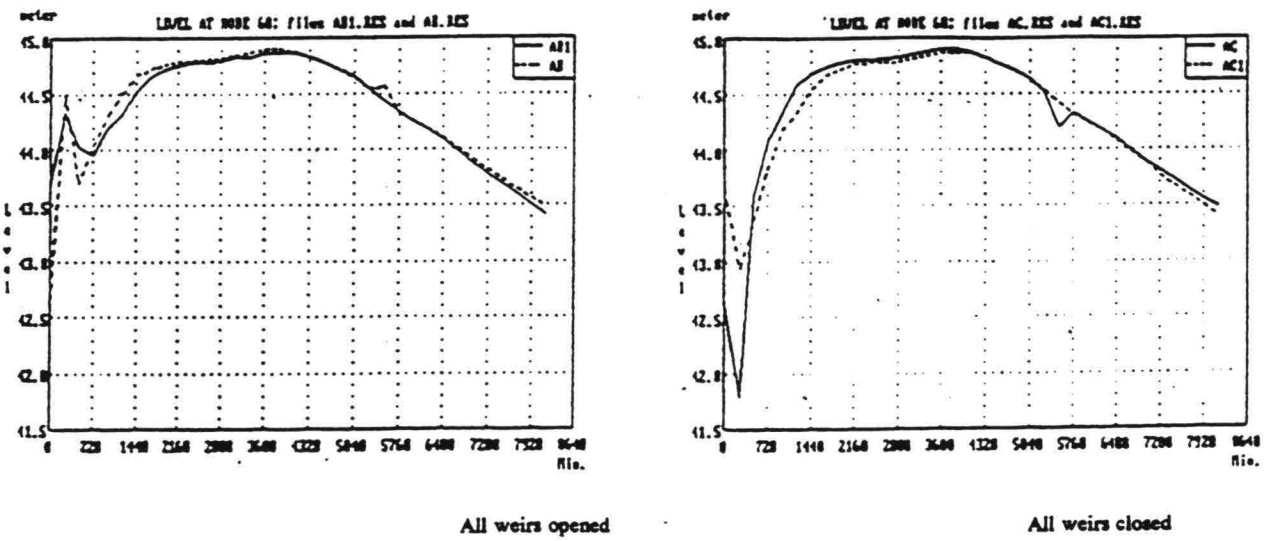
The weir at Monsin is operated such that the upstream water level is approximately constant. The operation procedure as such is not exactly known and only 24-hourly measurements of the water level are available (see dots in figure 4.13). Some attempts were made to reconstruct this structure operation procedure using trigger conditions available in duflow. The results of one of the test runs is shown in figure 4.13 indicating the computed water levels upstream Monsin and the line connecting the 24 hours level observations. Deviations are in the range of 30 cm with some peaks.



All weirs opened

All weirs closed

Figure 4.11 Levels at Monsin, simplified (——) and detailed (- - - - - -) model



All weirs opened

All weirs closed

Figure 4.12 Levels downstream Borgharen weir, simplified (——) and detailed (- - - - - - -) model

A special complication was that Monsin weir has a combined gate operation: overflow in low flow conditions and underflow in high flow conditions. Special care was taken for the transition from overflow to submerged flow condition.

The trigger conditions at Monsin weir for this test run are specified as:

No.	Trigger condition	Operation		
		H_{sill}	H_{gate} - H_{gate}	duration (hours)
0		55.18	∞	2
1	H > 57.70	52.58	53.50 - 53.75	2
2			53.75 - 54.00	4
3			54.00 - 54.50	4
4			54.50 - 55.00	4
5			55.00 - 55.50	4
6	H < 57.30		55.50 - 55.00	4
7			55.00 - 54.50	4
8			54.50 - 54.00	4
9			54.00 - 53.75	4
10			53.75 - 53.50	4
11	H < 57.20	55.18	∞	4

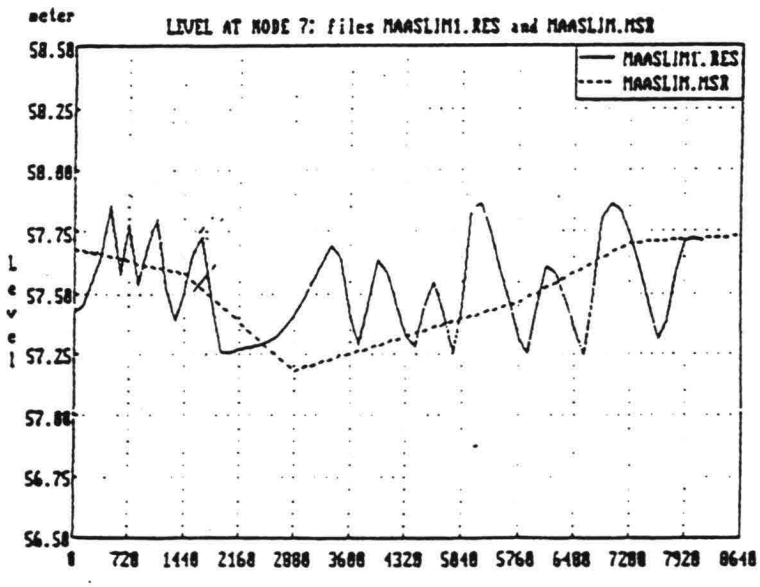


Figure 4.13 Simulated water levels upstream Monsin for a predefined trigger condition.

5. Conclusions

The duflow package has been extended for simulation of quality.

The model simulates now both the flow and water quality behaviour in channel networks. Version 1.0 has shown to provide accurate results for a variety of applications. Applications for which duflow is tested are: tidal driven flow, flood propagation in rivers, irrigation networks and systems of drainage canals. Other possible applications of the present model are oscillations and wind-driven circulation in harbours, lakes and lagoons. Version 2.0 has been tested on different types of systems too, but real world applications still have to be made.

The package is operational on the now widely available personal computers running under ms-dos. Despite the low requirements regarding computer capacity the program can handle networks of considerable size, up to 250 sections. With the file dufdim the user can estimate if the application can be handled by the program duflow, see Appendix A.

For simulation of water quality the number of applications is unlimited. As the user can define the process description in principle all types of models can be made. Including, oxygen, eutrophication, sediment transport and models describing fate and behaviour of heavy metals and organic micro pollutants.

A possible area of application not yet covered by the model is flow in sewers. Since closed channels (pipes) can be included without much difficulty, ideas have been developed to add a procedure for closed channels in the near future. This will extend the range of applications to water supply and sewerage systems, both in flat and inclined terrain.

Also from a water quality point of view this might be a very interesting option.

The contributors to the duflow package welcome any suggestions and potential contributions which may lead to improvement, updating and extension of the present package.

Interested organizations are kindly requested to reflect their ideas to the contributors mentioned at the front page of this manual or to contact Bureau Icim, The Hague, The Netherlands.

References

(not all references are mentioned in the text)

Abbott, M.B. (1979). Computational Hydraulics. Pitman.

Bulirsch R. & J. Stoer (1978). Einführung in die Numerische Mathematik. Springer.

Dronkers, J.J. (1964). Tidal Computations in Rivers and Coastal Waters. North Holland.

Jørgensen, S.E. & M.J. Gromiec (1989). Mathematical submodels in water quality systems.

Mahmood, K. & V. Yevyevich (1975). Unsteady Flow in Open Channels, Vol. I-II. Fort Collins, Water Resources Publ.

Roache, P.J. (1972). Computational Fluid Dynamics. Hemsha Publishers.

Thomann, R.V. & J.A. Mueller (1987). Principles of Surface Water Quality Modelling and Control. Harper international edition, New york.

Ven te Chow, P.D. (1983). Open-channel Hydraulics. Mc Graw-Hill.

Appendix A Installation of DUFLOW package and memory calculation

A.1. Program structure

The program is split up into a main program and three separate modules for input, calculations and result processing, which communicate by data files on the disk. The network file (extension .net) contains the network schematization, the boundary file (extension.bnd) holds all time dependent data like boundary conditions and control data.

The file CONFIG.DUF includes the configuration data and the names of the files which are currently in use.

A.2. Configuration requirements

- IBM-compatible computer with MS-DOS 640 Kb of memory
- Hard disk
- Mathematical coprocessor is advised but not necessary.
- Graphical card: CGA, EGA, VGA, OLIVETTI or HERCULES (or compatible).

On IBM PS/2: use "mode co80".

A.3. Installing the duflow package

Your duflow package includes the file setup.bat. Since the files on the distribution disks are compressed, you must use this program to install duflow.

Place the duflow installation diskette in your disk drive and type:

```
[origin:]SETUP <origin:> <destination_drive:> <destination_path>
```

Explanation

[origin:]	Drive in which the duflow installation diskette is in.
<destination_drive:>	Drive to install duflow to.
<destination_drive:>	Directory in which the duflow files are to be put.

For example when the duflow installation diskette is in drive a:, and when duflow is to be installed to c:\DUFLOW the command line will be:

```
a:setup a: c: \duflow
```

NOTE: the blank between [destination_drive:] and [destination_path:] is essential.

Installation on networks:

Use setup to copy the duflow system onto your network. Now the file config.Duf can be write-protected (if desired). Every user can call duflow from his own directory by a bath file duflow.bat:

```
@ECHO OFF
REM save the old path in environment variable
SET OLD=%PATH%

REM set new path and append
PATH C:\DOS;E:\APPLIC\DUFLOW;
APPEND E:\APPLIC\DUFLOW;

REM copy configuration file with default settings
IF NOT EXIST CONFIG.DUF COPY E:\APPLIC\DUFLOW\CONFIG.DUF

REM start program
E:\APPLIC\DUFLOW\DUFLOW

REM restore old situation
SET PATH=%OLD%
SET OLD=
APPEND;
```

To make screen dumps of the graphs the DOS command "graphics" must be executed before starting the program.

On machines with a hercules graphical card the command MSHERC must be executed before starting duflow.

Settings

To install the settings for your configuration choose "S =SETUP" from the "MASTER MENU". (See Paragraph 3.9)

A.4. Files

After installation the duflow directory will contain the following files:

duflow	.exe	Main program
setupdf	.exe	Setup program
iduflo	.exe	Input module
cduflo	.exe	Computational module
oduflo	.exe	Output module
dufdim	.exe	Program for estimating required memory (refer to section A.5)
duprol	.exe	DUFLOW compiler
newinits	.exe	Program for generating new initital conditions from a previous simulation
brun45	.exe	Additional files for programs
setupdf	.txt	
oduflo	.txt	
iduflo	.txt	
duflow	.txt	
duflow	.fon	
8x8	.fon	
8x16	.fon	
8x14	.fon	
dfsetup	.txt	
8x16sm	.fon	
6x6	.fon	

msherc	.com	Device driver file for HERCULES graphical card
config	.duf	DUFLOW configuration file.

SETUP will also create a subdirectory with additional examples, the project files (*.PRJ) of these examples are:

rizout	.prj	Demo example used in Appendix B: 'Getting started.
oschel	.prj	Example Eastern Scheldt, paragraph 4.2
sluice	.prj	Example Drainage Sluice, apragraph 4.3
meuse	.prj	Example Belgian Meuse, paragraph 4.4
eutrof1	.prj	Example eutrofication model eutrof1, appendix C
eutrof2	.prj	Example eutrofication model eutrof12, appendix D

A.5. Memory calculation

The size of the application made by the user is restricted. Like in duflow 1.00 a network up to 250 sections can be used.

In order to estimate the memory needed for a certain application the program dufdim can be used. This program, which is included on the diskette gives a rough estimation of the memory needed for the available computational options (flow, quality and all). Also an estimation of the maximum number of time steps to be calculated is provided.

dufdim is started by entering:

```
c:\duflow>dufdim
```

The program prompts the user:

Number of nodes:	number of nodal points in network
Number of sections:	number of sections in network
Number of structures:	number of weirs (overflow, underflow and culvert) in the network
Number of pumps:	number of pumps in the network
Number of siphons:	number of siphons in the network
Number of values/crosssection	number of depths to describe the cross-sectional profile used in the menu "INPUT -FLOW - NETWORK - CROSS-SECTIONS"
Modeldefinitionfile:	name of the *.mob file providing the process descriptions for quality simulation
Number of boundaries Flow:	number of boundaries used in menu "INPUT - FLOW - BOUNDARY CONDITIONS"
Number of boundaries Quality:	number of boundaries used in menu "INPUT - QUALITY - BOUNDARY CONDITIONS"
Number of output functions:	number of output functions selection in menu "INPUT - CONTROL DATA - QUALITY VARIABLES FOR OUTPUT" (in this are not included the state variables)

The number of words needed for the application

Memory needed	Flow
	Quality
	All

A maximum of 50.000 words of memory is available. If the estimated number of words is greater than 50.000, the application can not be simulated.

DUFDIM also calculates the maximum number of time steps.

The following two examples give an indication of the limits of the model.

In example 1 the conservative constituent `zout' is used in the application. In example 2 is the application EUTROF2 given (see Appendix D). A model with 11 water state variables and 8 bottom variables. The names of the model definition files in the examples are arbitrary chosen.

Example 1:

```
DUFDIM --- DUFLOW memory dimensioning utility v1.0
Number of nodes                  : 120
Number of sections               : 120
Number of structures             : 0
Number of pumps                  : 0
Number of siphons                : 0
Number of values / crosssection: 2
Modeldefinitionfile              : C:\DUFLOW\RIEUT1\EUTROF2.MOB
Number of boundaries Flow        : 50
                     Quality     : 50
Number of output functions       : 10
Memory needed Flow          11856 words              ( 50000 words available)
              Quality       32986 words
              All           33306 words
Max. number of timesteps Flow                                  48944
                         Quality                               41534
                         All              Flow                 45654
                                          Quality              41214
```

Example 2:

```
DUFDIM --- DUFLOW memory dimensioning utility v1.0
Number of nodes                  : 250
Number of sections               : 250
Number of structures             : 20
Number of pumps                  : 2
Number of siphons                : 2
Number of values / crosssection: 5
Modeldefinitionfile              : C:\DUFLOW\RIZOUT\ZOUT.MOB
Number of boundaries Flow        : 20
                     Quality     : 20
Number of output functions       : 0
Memory needed Flow          33084 words              ( 50000 words available)
              Quality       36896 words
              All           37226 words
Max. number of timesteps Flow                                  49214
                         Quality                               30323
                         All              Flow                 48593
                                          Quality              29993
```

The application can not be done if the needed memory is larger than the available memory or if the maximum number of time steps is equal to zero. With the maximum number of time steps calculated, the user can make an estimation of the maximum simulation time. For example in example 2 the maximum number of time steps for the calculation option All is 48500 for Flow and 30000 for

Quality. This means that, with a time step of 30 minutes for Flow and 1 hour for Quality, it gives at least a simulation time of 1000 days.

DUFDIM gives a worst case estimation. If the structure of the network is rather simple the memory needed can be much less than calculated with dufdim. In that case even if dufdim indicates that not enough memory is available, it can be worthwhile to try to run duflow.

Appendix B Getting started, an applied example

A simple problem is entered and computed in a step by step approach in this appendix.
First of all the program must be installed correctly (see Appendix A).
The program is stored at the directory DUFLOW.
The program is started up by typing: DUFLOW

```
C:\DUFLOW> DUFLOW
```

The program will show an introduction screen.
Press any key to continue
The program gives general information of DUFLOW.
Press any key to continue.
The program shows the MASTER MENU

```
DUFLOW 2.xx                    Master Menu
Data directory    : C:\DUFLOW\RIVER\
Project           : RIEUT1

                      MASTER MENU

                      I     Input
                      C     Calculations
                      O     Output

                      F     File names specification
                      M     quality Model development
                      S     Setup

                      Q     Quit
```

Give in the wanted names of the project.
choice: File name specification

```
DUFLOW  2.xx                   Filenames specification
 Data directory          : C:\DUFLOW\RIVER\
 Project                 : RIEUT1        (.PRJ)

 Control data                    : RIEUT1        (.CTR)
 FLOW
        Network data             : RIVER         (.NET .NOD)
        Initial conditions       : RIVER         (.BEG)
        Boundary conditions      : RIVER         (.BND)
        Results                  : RIVER         (.RES)
 QUALITY
        model                    : EUTROF1       (.MOB)
        Initial conditions       : RIEUT1        (.BEK)
        Boundary conditions      : RIEUT1        (.BNK)
        External variables       : RIEUT1        (.EXT)
        Parameters               : RIEUT1        (.PRM)
        Results                  : RIEUT1        (.REK)
```

Enter new path for data files
Enter new name (without extension) ↵ -No adjust = -Same
The programs returns to the MASTER MENU
choice: Input

```
DUFLOW 2.xx                    Input Menu
Data directory  : C:\DUFLOW\RIVER\
Project         : RIEUT1

                    MENU INPUT

                    C     Control data
                    F     Flowdata
                    Q     Quality data

                    S     Save as.. .
                    D     measured Data
```

Now the data of the network will be entered.
choice: Flow data

```
DUFLOW 2.xx                    Input - Flow - Menu
Data directory  : C:\DUFLOW\RIVER\   Network data          : RIVER.NET
Project         : RIEUT1             Initial conditions    : RIVER.BEG
                                     Bound.cond./ struc ctrl. : RIVER.BND

                    MENU INPUT - FLOW DATA

                    N     Network
                    I     Initial conditions
                    B     Boundary conditions
                    S     Structure control

                    M     Menu Input
```

choice: Network

```
DUFLOW 2.xx                    Input - Flow - Network - menu
Data directory  : C:\DUFLOW\RIVER\          Network data  : RIVER.NET
Project         : RIEUT1

                    MENU INPUT - FLOW - NETWORK

                    D     network Definition
                    N     Nodes
                    S     Sections
                    C     Cross-sections
                    U     strUctures

                    M     Menu inpu t - flow data
```

choice: network Definition

After entering the data of the nodes, the following screen will appear.

DUFLOW 2.xx — Input - Flow - Network - Definition

NETWORK DEFINITION — Press TAB to switch to node renumbering

Section/ structure	Begin Node	End Node
1	1	2
2	2	3
3	3	4
4	4	5
0		

Orientation of network:
0.00 ° from N to Y axis

The program returns to MENU INPUT - FLOWDATA - NETWORK
CHOICE: Nodes

DUFLOW 2.xx — Input - Flow - Network - Nodes

NODES general and flow related parameters

NODE	X-COORD. (m)	Y-COORD. (m)	CATCHMENT AREA (ha)	RUNOFF FACTOR
1	0	0	0.00	0.00
2	1000	0	0.00	0.00
3	2000	0	0.00	0.00
4	3000	0	0.00	0.00
5	4000	0	0.00	0.00
0				

The program returns to menu input - FLOW DATA - NETWORK
CHOICE: Sections

DUFLOW 2.xx — Input - Flow - Network - Sections

SECTIONS general and flow related parameters

Section	Length (m)	Direction °CW fr.N	Bottom Level (m) Begin	Bottom Level (m) End	Resistance Pos. dir.	(C or k) Neg. dir.	Windconv (10^-6)
1	1000.0	90.0	-2.0	-2.0	30.0	30.0	0.00
2	1000.0	90.0	-2.0	-2.0	30.0	30.0	0.00
3	1000.0	90.0	-2.0	-2.0	30.0	30.0	0.00
4	1000.0	90.0	-2.0	-2.0	30.0	30.0	0.00
0							

The program returns to MENU INPUT - FLOW DATA - NETWORK
CHOICE: Cross-sections
Enter the data for the first section

DUFLOW 2.xx — Input - Flow - Network - Cross-sections

CROSS-SECTION Profile SECTION

Depth to bott.(m)	Flow width (m) at begin	Flow width (m) at end	Storage Width (m) at begin	Storage Width (m) at end
0.00	10.00	10.00	10.00	10.00
2.00	10.00	10.00	10.00	10.00

The program returns to MENU INPUT - FLOW DATA - NETWORK
CHOICE: Esc, because there are no structures in this problem.

The program returns to menu input - FLOW DATA
CHOICE: Initial conditions

```
DUFLOW 2.xx                    Input - Flow - Initial cond. - Menu
Data directory  : C:\DUFLOW\RIVER\  Initial conditions      : RIEUT1.BEG
Project         : RIEUT1

                    MENU INPUT - FLOW DATA - INITIAL CONDITIONS

                    H     initial levels
                    Q     initial discharges

                    M     Menu input - flow data
```

CHOICE: H initial levels

```
DUFLOW 2.xx                    Input - Flow - Initial cond. - Levels

INITIAL LEVELS
| NODE | Level (m) |
|------|-----------|
|    1 |      0.00 |
|    2 |      0.00 |
|    3 |      0.00 |
|    4 |      0.00 |
|    5 |      0.00 |
```

The program returns to menu input - flow data - initial conditions
CHOICE: Q initial discharges

```
DUFLOW 2.xx                    Input - Flow - Initial cond. - Discharges
INITIAL DISCHARGES, sections
| SECTION | Discharge (m3/s) |       |
|         | Begin            | End   |
|---------|------------------|-------|
|       1 |            1.000 | 1.000 |
|       2 |            1.000 | 1.000 |
|       3 |            1.000 | 1.000 |
|       4 |            1.000 | 1.000 |
```

The program returns to MENU INPUT - FLOW DATA with Esc
CHOICE: Boundary conditions

```
DUFLOW 2.xx                    Input - Flow - Boundary cond.
FLOW BOUNDARY CONDITION Nr 1
Type              : Q add.                          Unit: Q^3/s
Node              : 1
Type of function  : Constant
Value             : 1.00000
```

After the second condition, in which the level at node 5 is put as a constant to zero.
CHOOSE: None erase.
The program returns to MENU INPUT - FLOW DATA

No structures defined, so there is no need for entering structure control
CHOICE: Esc

The program returns to MENU INPUT
Before entering the quality data, the variables must be defined already.
CHOICE: Esc

The program prompts, if any changes are made:
Do you want to save the input data (Y/N?)
CHOICE: **Y**

The program returns to MASTER MENU
CHOICE: quality Model development

```
DUFLOW 2.xx                    Quality model develop ment
Data directory : C:\DUFLOW\RIVER\
Selected model file: EUTROF1.MOB
                          MENU QUALITY MODEL DEVELOPMENT

                          E      Edit
                          C      Compile

                          M      Master Menu
```

CHOICE: Edit
Then the program will ask: Model definition file to be edited (eutrof1.mod)
The editor will be activated so the desired process description file can be entered. The result of the editing is shown below:

```
/*      Simple Eutrophication Model  EUTROF1.MOD  DUFLOW v2.xx                  */
/*                                                                             */
/*      Hans Aalderink & Nico Klaver                                           */
/*                                                                             */
/*                      Agricultural University of Wageningen                  */
/*                      Department of Nature Conservation                      */
/*                      Water Quality Management Section                       */
/*                      P.O. BOX 8080                                          */
/*                      6700 DD Wageningen                                     */
/*                      The Netherlands                                        */
/*                                                                             */
/*      September 1992                                                         */

water   A       [ 2.000] mg-C/l         ;Algal biomass
water   PORG    [ 0.110] mg-P/l         ;Organic Phosphorus
water   PANORG  [ 0.040] mg-P/l         ;Inorganic Phosphorus
water   NH4     [ 0.300] mg-N/l         ;Ammonia
water   NO3     [ 3.000] mg-N/l         ;Nitrate
water   NORG    [ 0.800] mg-N/l         ;Organic Nitrogen
water   O2      [10.000] mg/l           ;Oxygen
water   BOD     [ 5.000] mg-O2/l        ;BOD-5
water   SS      [ 5.000] mg/l           ;Suspended Solids
parm    kp      [ 0.005] mg-P/l         ;Monod constant Phosphorus
parm    kn      [ 0.010] mg-N/l         ;Monod constant Nitrogen
parm    ealg    [ 0.016] ug-Chl/l,m     ;Specific extinction chlorophyll
parm    e0      [ 1.000] 1/m            ;Background extinction
parm    achlc   [30.000] ug-Chl/mg-C    ;Chlorophyll to Carbon ratio
parm    is      [40.000] W/m2           ;Optimal Light Intensity
```

```
parm    umax        [ 4.000] 1/day          ;Unlimited algal growth rate
parm    tga         [ 1.047] -              ;Temperature coefficient algal growth
parm    tra         [ 1.047] -              ;Temp. coefficient algal respiration
parm    kres        [ 0.100] 1/day          ;Respiration rate constant
parm    kdie        [ 0.200] 1/day          ;Die rate constant
parm    fporg       [ 1.000] -              ;Fraction PORG released by respiration
parm    apc         [ 0.025] mg-P/mg-C      ;Phosphorus to Carbon ratio
parm    anc         [ 0.250] mg-N/mg-C      ;Nitrogen to Carbon ratio
parm    kmin        [ 0.100] 1/day          ;Rate constant mineralisation
parm    tmin        [ 1.047] -              ;Temperature coefficient mineralisation
parm    vso         [ 0.100] m/day          ;Nett sedimentation rate organic matter
parm    fdporg      [ 0.000] -              ;Fraction dissolved organic Phosphorus
parm    fdnorg      [ 0.000] -              ;Fraction dissolved organic Nitrogen
parm    kpip        [ 0.010] l/mg-SS        ;Phosphorus Partition coefficient
parm    fnorg       [ 1.000] -              ;Fraction NORG released by respiration
parm    kmn         [ 0.025] mg-N/l         ;Ammonia preference factor
parm    tnit        [ 1.080] -              ;Temperature coefficient nitrification
parm    knit        [ 0.100] 1/day          ;Nitrification rate constant
parm    kno         [ 2.000] mg-O2/l        ;Monod constant nitrification
parm    kden        [ 0.100] 1/day          ;Denitrification rate constant
parm    tden        [ 1.045] -              ;Temperature coefficient denitrification
parm    kdno        [ 0.500] mg-O2/l        ;Monod constant denitrification
parm    kbod        [ 0.100] 1/day          ;Oxidation rate constant BOD
parm    tbod        [ 1.047] -              ;Temperature coefficient oxidation BOD
parm    kbodo       [ 2.000] mg-O2/l        ;Monod constant oxidation BOD
parm    fdbod       [ 1.000] -              ;Fraction dissolved BOD
parm    aoc         [ 2.670] mg-O2/mg-C     ;Oxygen to Carbon ratio
parm    trea        [ 1.024] -              ;Temperature coefficient reaeration
parm    krmin       [ 0.100] m/day          ;Minimum oxygen mass transfer constant
parm    vss         [ 0.100] m/day          ;Sedimentation rate Suspended Solids

xt      sod         [ 1.000] g-O2/m2.day    ;Sediment Oxygen Demand
xt      i0          [ 10.00] W/m2           ;Surface Light Intensity
xt      t           [ 20.00] oC             ;Temperature
xt      resf        [  0.50] g/m2.day       ;Resuspension flux Suspended Solids
xt      pflux       [  0.00] g P/m2,day     ;Phosphorus release flux from sediment
xt      nflux       [  0.00] g N/m2,day     ;Ammonia release flux from sediment

flow    z           [  2.00] m              ;Water depth
flow    Q           [  0.10] m3/day         ;Flow
flow    As          [ 10.00] m2             ;Cross sectional Area

{

fdpano=1/(1+kpip*SS);
PORTO=PANORG*fdpano;
Chla=achlc*A;

fn=MIN(PORTO/(PORTO+kp),(NH4+NO3)/(NH4+NO3+kn));
etot=e0+ealg*Chla;
ister=i0/is;
fl=2.71*(exp(-1*ister*exp(-1*etot*z))-exp(-ister))/(etot*z);
ft=tga^(t-20);
Groei=umax*fn*fl*ft;
Resp=kres*tra^(t-20)+kdie;
k1(A)=Groei-Resp;

mino=kmin*tmin^(t-20);
```

```
sedo=vso/z;

k0(PORG)=fporg*Resp*apc*A;
k1(PORG)=-1*mino-sedo*(1-fdporg);

k0(PANORG)=mino*PORG-Groei*A*apc+(1-fporg)*Resp*apc*A+pflux/z;
k1(PANORG)=-1*vss/z*(1-fdpano);

k0(NORG)=fnorg*Resp*anc*A;
k1(NORG)=-1*mino-1*sedo*(1-fdnorg);

if ((NO3==0.0) && (NH4==0.0))
        {
        pnh4=0.;
        }
else
        {
        pnh4=NH4*NO3/((kmn+NH4)*(kmn+NO3))+NH4*kmn/((NH4+NO3)*(kmn+NO3)) ;
        }

nitr=knit*tnit^(t-20)*O2/(O2+kno);
k0(NH4)=mino*NORG-Groei*anc*A*pnh4+(1-fnorg)*Resp*anc*A+nflux/z;
k1(NH4)=-1*nitr;

denit=kden*tden^(t-20)*kdno/(kdno+O2);
k0(NO3)=nitr*NH4-Groei*anc*A*(1-pnh4);
k1(NO3)=-1*denit;

oxid=kbod*tbod^(t-20)*O2/(O2+kbodo);
conv=(1-exp(-5*kbod));
k1(BOD)=-1*oxid-1*sedo*(1-fdbod);
k0(BOD)=(kdie*aoc*A-5/4*32/14*denit*NO3)*conv;

u=ABS(Q/As);
kmas=3.94*u^0.5*z^(-0.5);
if (kmas<krmin)
        {
        kmas=krmin;
        }
kre=(kmas*trea^(t-20))/z;
cs=14.5519-0.373484*t+0.00501607*t*t;
k1(O2)=-1.0*kre;
k0(O2)=kre*cs-oxid*BOD/conv-64/14*nitr*NH4-32/12*kres*tra^(t-20)*A-sod/z+Groei*A*
(32/12+48/14)*anc*(1-pnh4)*NO3;

k1(SS)=-1*vss/z;
k0(SS)=resf/z;

Ptot=PORG+PANORG+A*apc;
Nkj=NORG+NH4+anc*A;
Ntot=Nkj+NO3;}
```

After editing the model file, leave the editor
The program will return to QUALITY MODEL DEVELOPMENT MENU
CHOICE: Compile
The program translate the EUTROF1.MOD file into a EUTROF1.MOB file.
The program returns to QUALITY MODEL DEVELOPMENT MENU

CHOICE: Esc to return to MASTER MENU

CHOICE: Input
CHOICE: Quality

DUFLOW 2.xx — Input - Quality - Menu

Data directory	: C:\DUFLOW\RIVER\	Initial conditions	: RIEUT1.BEK
Project	: RIEUT1	Boundary conditions	: RIEUT1.BNK
Model	: EUTROF1.MOB	External variables	: RIEUT1.EXT
		Parameters	: RIEUT1.PRM

MENU INPUT - QUALITY DATA

I Initial conditions
B Boundary conditions
E External variables
P Parameters

M Menu Input

CHOICE: Initial conditions
Make a choice from the checklist, see below:

DUFLOW 2.xx — Input - Quality - Initial cond.

Quality variables

Check	Variable	Description
√	a	algal biomass
√	bod	bod-5 biomass
.	nh4	ammmonia
.	no3	nitrate
.	norg	organic nitrogen
√	o2	oxygen
.	panorg	organic phosphorus
.	porg	organic phosphorus
.	ss	suspended solids

Three state variables of the checklist are selected now.
Press return to edit or look the specific values.
The screen will first display the default values, by editing the values can changed.

DUFLOW 2.xx — Input - Quality - Initial cond.

INITIAL CONDITIONS

NODE	a mg-c/l	bod mg-o2/l	o2 mg/l
1	2.000	5.000	10.000
2	2.000	5.000	10.000
3	2.000	5.000	10.000
4	2.000	5.000	10.000
5	2.000	5.000	10.000

The default values are not changed.
CHOICE: Esc to return to MENU INPUT - QUALITY DATA
CHOICE: Boundary conditions

DUFLOW 2.xx Input - Quality - Boundary conditions

DEFINED QUALITY BOUNDARY CONDITION

Check	Type	Variable	Node(s)
.	Concentration	a	1
.	Concentration	nh4	1
.	Concentration	no3	1
.	Concentration	norg	1
√	Concentration	o2	1
.	Concentration	panorg	1
.	Concentration	porg	1
.	Concentration	ss	1
√	Concentration	bod	1

After choosing the desired variables, with Enter you can edit the boundary conditions:

DUFLOW 2.xx Input - Quality - Boundary cond.

QUALITY BOUNDARY CONDITIONS Nr 1:

Type	: Concentration	
Variable	: o2	Unit : mg/l
Node	: 1	
Type of function	: Constant	
Constant value	: 10.000	

After entering the first boundary, a second one is entered:
CHOICE: return

DUFLOW 2.xx Input - Quality - Boundary cond.

QUALITY BOUNDARY CONDITIONS Nr 2:

Type	: Concentration	
Variable	: bod	Unit : mg-o2/l
Node	: 1	
Type of function	: Constant	
Constant value	: 2.000	

After entering the last boundary condition press Esc or return to the MENU INPUT - QUALITY DATA.
The program prompts:
Changes have been made in boundary conditions. Save to file? (Y/N).
Press Y so the entered data will be stored.

CHOICE: External variable
On the screen appears a checklist, of which the external variables can be chosen. For example we choose d, the dispersion coefficient.

DUFLOW 2.xx Input - Quality - External variables

EXTERNAL VARIABLE SPECIFICATION Nr.
external variable : d Unit : m2/s

Node(s) : A
Type of function : Constant
Constant value : 25

CHOICE: Esc to return to MENU INPUT - QUALITY DATA
CHOICE: Parameters

DUFLOW 2.xx Input - Quality - Parameters

Model parameters

Name	Description	Default	VALUE	unit
achlc	chlorophyll to carbon ratio	30.000	30.000	ug-chl/mg-
anc	nitrogen to carbon ratio	0.25000	0.10000	mg-n/mg-c
aoc	oxygen to carbon ratio	2.6700	2.670	mg-o2/mg-c
apc	phosphorus to carbon ratio	0.02500	0.01000	mg-p/mg-c
e0	background extinction	1.000	2.000	1/m
ealg	specific extinction chlorophyll	0.01600	0.01700	ug-chl/l,m
fdbod	fraction dissolved bod	1.000	1.000	-
fdnorg	fraction dissolved organic nitrogen	0.000	0.40000	-
fdporg	fraction dissolved organic phosphorus	0.000	0.000	-
fnorg	fraction norg released by respiration	1.000	0.6000	-
fporg	fraction porg released by respiration	1.000	0.60000	-
is	optimal light intensity	40.000	80.000	w/m2
kbod	oxidation rate constant bod	0.10000	0.10000	1/day
kbodo	monod constant oxidation bod	2.000	2.000	mg-o2/l
kden	denitrification rate constant	0.10000	0.09000	1/day
kdie	die rate constant	0.20000	0.2000	1/day
kdno	monod constant denitrification	0.50000	0.10000	mg-o2/l
kmin	rate constant mineralisation	0.10000	0.20000	1/day
kmn	ammonia preference factor	0.02500	0.02500	mg-n/l
kn	monod constant nitrogen	0.01000	0.02500	mg-n/l
knit	nitrification rate constant	0.10000	0.09000	1/day
kno	monod constant nitrification	2.000	2.000	mg-o2/l
kp	monod constant phosphorus	0.00500	0.00500	mg-p/l
kpip	phosphorus partition coefficient	0.01000	0.01000	l/mg-ss
kres	respiration rate constant	0.10000	0.12500	1/day
krmin	minimum oxygen mass transfer constant	0.10000	0.20000	m/day
tbod	temperature coefficient oxidation bod	1.047	1.047	-
tden	temperature coefficient denitrificati	1.045	1.045	-
tga	temperature coefficient algal growth	1.047	1.060	-
tmin	temperature coefficient mineralisatio	1.047	1.080	-
tnit	temperature coefficient nitrification	1.080	1.080	-
tra	temperature coefficient algal respira	1.047	1.045	-
trea	temperature coefficient reaeration	1.024	1.040	-
umax	unlimited algal growth rate	4.000	2.100	1/day
vso	nett sedimentation rate organic matte	0.10000	0.50000	m/day
vss	sedimentation rate suspended solids	0.10000	1.000	m/day

After editing the actual values, press Esc.
The quality data have been entered now.

CHOICE: Menu input to return to MENU INPUT
CHOICE: Control data to enter general data of the model

```
DUFLOW 2.xx                    Input - Control data - Menu
Data directory   : C:\DUFLOW\RIVER\   Control data    : RIEUT1.CTR
Project          : RIEUT1

                    MENU INPUT - CONTROL DATA

                    C     Calculation definition
                    L     Locations for output
                    Q     Quality variables for output
                    S     Special control data

                    M     Menu input
```

CHOICE: Calculation definition
After entering the calculation screen, the result is shown below:

```
DUFLOW 2.xx                    Input - Control data - Calc. definition
CALCULATION DEFINITION
IDENTIFICATION TEXT: EXAMPLE TO GET STARTED WITH DUFLOW
TIMES                      | yymmdd  | hhmm |
 Start of computation      : 910701    0000
 Start of output           : 910701    0000
 End of computation        : 910703    0000
 Time step Flow            :           0010     # Time steps  = 288
 Time step Quality         :           0010     # Time steps  = 288
 Output interval           :           0030     # Output steps =  96
HYDRAULIC CALCULATION----------------------
 Resistance formula        : De Chezy
 calc. of advection term   : Total
 Extra iteration           : No
 Theta          (0.5-1.0): 0.55
QUALITY CALCULATION------------------------
 Decouple                  : No
 Theta          (0.0-1.0): 0.55
```

The programs returns to MENU INPUT - CONTROL DATA
CHOICE: Locations for output

```
DUFLOW 2.xx                    Input - Control data - Locations

MONITORING OUTPUT
 Selected sections/struct.: 1-4

OUTPUT LOCATIONS
 Selected sections/struct.: 1-4
```

The programs returns to MENU INPUT - CONTROL DATA
CHOICE: Quality variables for output

DUFLOW 2.xx Input - Control data - Special

OUTPUT DEFINITION - QUALITY VARIABLES

Check	Variable	Description
✓	a	algal biomass
✓	bod	bod-5 biomass
.	nh4	ammmonia
.	no3	nitrate
.	norg	organic nitrogen
✓	o2	oxygen
.	panorg	organic phosphorus
.	porg	organic phosphorus
.	ss	suspended solids

The programs return to menu input - control data
choice: Special Control Data

DUFLOW 2.xx Input - Control data - Special

SPECIAL CONTROL DATA

Create intermediate flow file	: Yes
Alpha (corr. for velocity distribution)	: 1
Minimum # timesteps between triggers	: 3

Return to master menu by pressing Esc.
The program prompts:
Changes have been made in control data. Save to file? (Y/N).
Answer Y to store the control data.

CHOICE: Calculations
The MENU CALCULATIONS is shown below:

DUFLOW 2.xx Calculations

Data directory : C:\DUFLOW\RIVER\
Project : RIEUT1

MENU CALCULATIONS

A	All
F	Flow
Q	Quality
B	Box
M	Master Menu

CHOICE: All
The program starts the computations for both flow and quality.
After ending the computations the program prompts:
Calculations finished successfully.

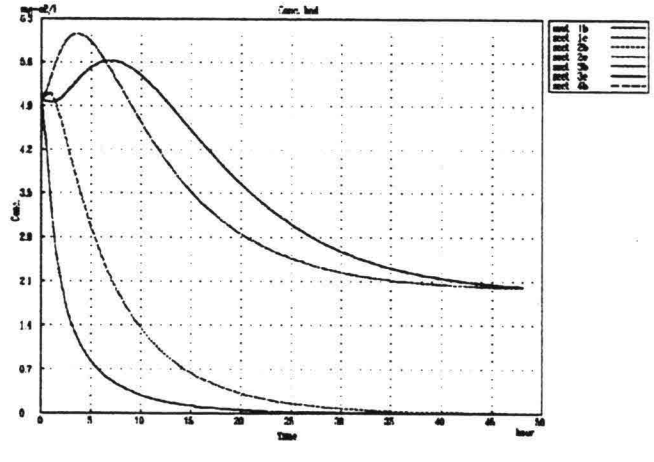
CHOICE: Master Menu
CHOICE: Output
The program will display menu output, like below:

```
DUFLOW 2.xx                    Output - Menu
Data directory  : C:\DUFLOW\RIVER\     Flow results      : RIVER.RES
Project         : RIEUT1               Qualityresults    : RIEUT1.REK

                          MENU OUTPUT

                          T  Time related output
                          V  time rel. - Various loc.

                          S  Space related output
                          R  definition Routes

                          M  Master menu
```

CHOICE: Time related output
The program will display the following table to be edited:

```
DUFLOW 2.xx                    Output -time related
SELECTION OF DATA
 nr.  type      variable   file
   1  Conc.      o2        RIEUT1.REK
   2  Conc.      bod       RIEUT1.REK

Sections/Structures     : 1B,1E,2B,2E,3B,3E,4B,4E
grouped together        : Locations
```

CHOICE: Graph
CHOICE: Screen



The graph for the variable O_2 will follow after this graph when Enter is pressed.

CHOICE: definition **Routes**
The program will display the following table to be edited:

DUFLOW 2.xx Output - definition routes

Route nr :
Route name :

Section		X-begin	X-end		node beg	node end
1		0	1000		1	2
2		1000	2000		2	2
3		2000	3000		3	4
4		3000	4000		4	5
0						

CHOICE: **Space related**
The program will display the following table to be edited:

DUFLOW 2.xx Output - space related

SELECTION OF DATA

nr.	type	variable	route	file
1	Conc.	bod	LANGS	RIEUT1.REK
2	Conc.	o2	LANGS	RIEUT1.REK

first output	: 910301	0000		
last output	: 910303	0000		
interval	: 00	0600	# timepoints:	8
grouped together	: Variables			

CHOICE: **Table**
The program will display the beginning of the table on the screen.

Conc. 1 91/07/01 00:00

Length meter	bod mg-o2/l	o2 mg/l
0.000	5.000	10.000
1000.0	5.000	10.000
1000.0	5.000	10.000
2000.0	5.000	10.000
2000.0	5.000	10.000
3000.0	5.000	10.000
3000.0	5.000	10.000
4000.0	5.000	10.000

Conc. 1 91/07/01 06:01

Length meter	bod mg-o2/l	o2 mg/l
0.000	0.65676	8.291
1000.0	2.524	9.170
1000.0	2.524	9.170
2000.0	5.716	9.868
2000.0	5.716	9.868
3000.0	5.601	10.127
3000.0	5.601	10.127
4000.0	5.294	10.163

Appendix C EUTROF1

EUTROF1 is one of the two pre-defined eutrophication models included in DUFLOW. It is a relatively simple model based on the us epa model eutro4. It includes the cycling of nitrogen, phosphorus and oxygen. The growth of one phytoplankton species is simulated. The interaction between the sediment and the overlying water column is not included in a dynamic way. Sediment exchange fluxes of oxygen, ammonia and phosphorus can be specified by the user. These fluxes may be location specific and time dependent, reflecting temporal and seasonal variations.

The model is in particular suitable to study the short term behaviour of systems. For example to examine the impacts of a discharge on the oxygen dynamics, or to explore the effects of flushing on the chlorophyll-a concentration.

In case the long term functioning of a system is of interest the other pre-defined eutrophication model EUTROF2 is more appropriate. EUTROF2 includes three algal species, so succession can be simulated to a certain extend. Furthermore this model also describes the interactions between the sediment and the overlying water column.

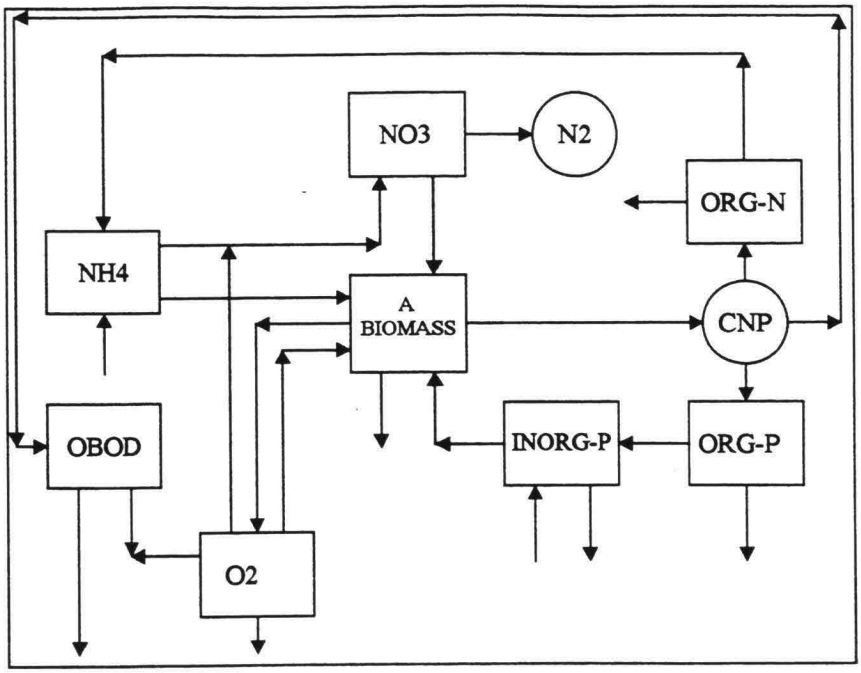
State variables
Figure C-1 presents the principle kinetic interactions for the modelled state variables. The model includes the following state variables:

- A Algal Biomass (mg C/l)
- Porg Organic Phosphorus (mg P/l)
- Pinorg Inorganic Phosphorus (mg P/l)
- Norg Organic Nitrogen (mg N/l)
- NH4 Ammonia Nitrogen (mg N/l)
- NO3 Nitrate Nitrogen (mg N/l)
- O2 Oxygen (mg O_2/l)
- bod Carbon 5 day Biochemical Oxygen Demand (mg O_2/l)
- SS Suspended Solids (mg/l)

Besides the state variables mentioned in figure C-1 a number of output variables are calculated:

- Porto Dissolved Inorganic Phosphorus (mg P/l)
- Ptot Total Phosphorus (mg P/l)
- Nkj Kjeldahl- Nitrogen (mg N/l)
- Ntot Total Nitrogen (mg N/l)
- Chl-a Chlorophyll-a ((g/l)

These variables are often monitored, so a direct comparison between measurements and simulated results is possible for these constituents.



Figuur C-1 EUROTROF1 State variabele interactions

Process descriptions

In the following part the equations, describing the processes are presented. The explanation of the symbols used in this part is given in tables C-1 and C-2 at the end of Appendix C

- **Algae**

$$\frac{dA}{dt} = \left[\mu_{max} F_T F_N F_I\right] A - \left[k_{res}\theta_{ra}^{(T-20)} + k_{die}\right] A \qquad \text{(C-1)}$$

Algal growth is considered to be limited by nutrients, light and temperature. Nutrient limitation is described as:

$$F_N = \min\left[\frac{P_{ortho}}{P_{ortho} + K_r}, \frac{(NH_4 + NO_3)}{(NH_4 + NO_3) + K_N}\right] \qquad \text{(C-2)}$$

Which means that the reduction of the maximum growth rate is controlled by the most limiting factor. It is assumed that algae can use both ammonia and nitrate for their growth. The uptake of both nitrogen constituents is controlled by the ammonia preference factor (see eq. C-11).

Light limitation is described using the depth averaged Steele equation:

$$F_I = \frac{e}{\varepsilon_{tot} z}\left[\exp\left(-\frac{I_o}{I_s}\exp(-\varepsilon_{tot} z)\right) - \exp\left(-\frac{I_o}{I_s}\right)\right] \qquad \text{(C-3)}$$

in which:

$$\varepsilon_{tot} = \varepsilon_0 + \varepsilon_{alg} Chl - a + \varepsilon_{SS} SS \qquad \text{(C-4)}$$

Temperature dependency of the algal growth rate is given by:

$$F_T = \theta_{ga}^{(T-20)} \tag{C-5}$$

Temperature dependency for all process rates is described in the same way as in eq. C-5. The rate constant at a reference temperature of 20 °C is multiplied with a coefficient, determining the change per °C difference from the reference temperature.

Two terms describing the loss processes complete the algal balance equation. The first one describes the endogenous respiration, which is considered to be temperature dependent. The second is a lumped rate constant including death rate and the effect of grazing.

For internal computational purpose algal carbon is used as a measure for the biomass. The algal-C concentration is converted to chlorophyll-a using a fixed chlorophyll to carbon ratio.

- **Organic Phosphorus**

$$\frac{dP_{org}}{dt} = -k_{\min}\theta_{\min}^{(T-20)}P_{org} - \frac{v_{SO}}{z}(1-f_{dporg})P_{org} + f_{porg}\left[k_{res}\theta_{ra}^{(T-20)} + k_{die}\right]a_{pc}A \tag{C-6}$$

During the phytoplankton loss processes part of the associated phosphorus is released as organic phosphorus, the remaining part is distributed to the inorganic phosphorus pool. The phosphorus to carbon ratio is assumed to be constant. Due to mineralisation, organic phosphorus is converted to inorganic phosphorus. Mineralisation is described as a temperature dependent process. Part of the organic phosphorus is present in a particulate form and is subject to settling.

- **Inorganic Phosphorus**

$$\frac{dP_{anorg}}{dt} = -\frac{v_{SS}}{z}(1-f_{dpano})P_{anorg} + k_{\min}\theta_{\min}^{(T-20)}P_{org} - \mu_{\max}F_T F_N F_l a_{pc}A + (1-f_{porg})\left[k_{res}\theta_{ra}^{(T-20)} + K_{die}\right]a_{pc}A + \frac{P_{flux}}{z} \tag{C-7}$$

Inorganic phosphorus will be formed during mineralisation of organic phosphorus and is also released during the algal respiration and die-off. Part of the inorganic phosphorus is adsorbed to the suspended solids. The dissolved fraction is calculated using:

$$f_{dpano} = \frac{1}{1+K_{pip}SS} \tag{C-8}$$

The use of eq. C-8 implies that it is assumed that the equilibrium is reached instantaneously. The sorption rate is fast compared to most other relevant processes. Furthermore it is assumed that the linear part of the sorption isotherm may be used.

- **Organic Nitrogen**

$$\frac{dN_{org}}{dt} = -k_{\min}\theta_{\min}^{(T-20)}N_{org} - \frac{v_{so}}{z}(1-f_{dnorg})N_{org}$$

$$+f_{norg}\left[k_{res}\theta_{ra}^{(T-20)}+k_{die}\right]a_{nc}A \qquad \text{(C-9)}$$

Organic nitrogen is produced during respiration and die-off of algae. Like organic phosphorus part of the organic nitrogen is associated with particulate matter and will be subject to settling. Due to mineralisation organic nitrogen will be released as ammonia. De rate constant for the mineralisation of both organic nitrogen and phosphorus are assumed to be equal.

- **Ammonia Nitrogen**

$$\frac{dNH_4}{dt}=-k_{nit}\theta_{nit}^{(T-20)}\frac{O_2}{O_2+K_{NO}}NH_4+k_{\min}\theta_{\min}^{(T-20)}N_{org}-\mu_{\max}F_TF_NF_la_{nc}P_{NH4}A$$
$$+(1-f_{norg})\left[k_{res}\theta_{ra}^{(T-20)}+k_{die}\right]a_{nc}A+\frac{N_{flux}}{z} \qquad \text{(C-10)}$$

During the algal respiration and die-off part of the nitrogen included in the biomass is released as ammonia. The remaining part is added to the pool of organic nitrogen. Both ammonia and nitrate can be used for algal growth.

The preference for the nitrogen source is controlled by the preference factor given by:

$$P_{NH4}=NH_4\frac{NO_3}{(K_{mN}+NH_4)(K_{mN}+NO_3)}$$
$$+NH_4\frac{K_{mN}}{(NH_4+NO_3)+(K_{mN}+NO_3)} \qquad \text{(C-11)}$$

The nitrification rate is controlled by the oxygen concentration, using a Monod type of equation. Depending on the value of K_{NO} the rate can be limited at low oxygen concentrations.

- **Nitrate Nitrogen**

$$\frac{dNO_3}{dt}=-k_{den}\theta_{den}^{(T-20)}\frac{K_{dNO}}{K_{dNO}+O_2}NO_3+k_{nit}\theta_{nit}^{(T-20)}\frac{O_2}{O_2+K_{NO}}NH_4$$
$$-\mu_{\max}F_TF_NF_la_{nc}(1-P_{NH4})A \qquad \text{(C-12)}$$

Nitrate is formed during nitrification. Depending on the ammonia preference factor, nitrate can be used for algal growth. Denitrification, which is also controlled by the oxygen concentration is included too.

- **BOD 5**

$$\frac{dBOD}{dt}=-k_{BOD}\theta_{BOD}^{(T-20)}\frac{O_2}{O_2+K_{BODO}}BOD-\frac{v_{so}}{z}(1-f_{dBOD})BOD$$
$$+k_{die}a_{oc}A-\frac{5}{4}\frac{32}{14}k_{den}\theta_{den}^{(T-20)}\frac{K_{dNO}}{K_{dNO}+O_2}\Bigg]X_{conv} \qquad \text{(C-13)}$$

Eq. C-13 describes the 5 day carbon BOD, which is also used for input. In the oxygen balance equation (eq. C-15) the ultimate bod is used. A conversion factor is used to calculate BOD_5 from bodu. The bod, which is produced by die off of the algae is converted to BOD_5. Also the bod used as a carbon source during denitrification is corrected this way.

$$X_{conv} = 1 - \exp\left(-5k_{BOD}^{20}\right) \tag{C-14}$$

Part of the BOD is in a particulate form and will settle. The oxidation of BOD is temperature dependent and is limited at low oxygen concentrations by a Monod type of kinetic.

- **Oxygen**

$$\begin{aligned}\frac{dO_2}{dt} = {} & k_{re}\theta_{re}^{(T-20)}\left(c_s - O_2\right) - k_{BOD}\theta_{BOD}^{(T-20)}\frac{O_2}{O_2 + K_{BODO}}\frac{BOD}{X_{conv}} \\ & -\frac{64}{14}k_{nit}\theta_{nit}^{(T-20)}\frac{O_2}{O_2 + K_{NO}}NH_4 - \frac{32}{12}k_{res}\theta_{ra}^{(T-20)}A - \frac{SOD}{z} \\ & +\mu_{max}F_T F_N F_l A\left(\frac{32}{12} + \frac{48}{14}a_{nc}(1 - P_{NH4})NO_3\right)\end{aligned} \tag{C-15}$$

The mass transfer coefficient for oxygen is given by the following empirical equation:

$$k_{mas} = 3.94u^{0.5}z^{-1.5} \tag{C-16}$$

or if $k_{mas} < k_{rmin}$:

$$k_{mas} = k_{rmin} \tag{C-17}$$

The dimension of the velocity is m/s and the resulting k_{mas} is in m/day. At low stream velocity the use of eq. C-16 can result into extremely low value for the mass transfer coefficient. The user can define a minimum value for k_{mas}, which is used as a lower bound for the mass transfer coefficient. The reaeration rate constant kre is given by:

$$k_{re} = \frac{k_{mas}}{z} \tag{C-18}$$

The oxygen saturation concentration is also calculated, using an empirical equation:

$$C_S = 14.5519 - 0.373484T + 0.00501607T^2 \tag{C-19}$$

Additional to the oxidation of carbon bod the following oxygen consuming processes are included in the oxygen mass balance equation: the algal respiration, nitrification and the sediment oxygen demand. sod can be supplied by the user as a time dependent and location specific function, in order to simulate seasonal and temporal variations. Production of oxygen results from primary production. In case nitrate is used as a source for nitrogen an additional oxygen production takes place, because of the reduction of nitrate during the assimilation process.

- **Suspended Solids**

$$\frac{dSS}{dt} = -\frac{v_{SS}}{z}SS + \frac{\Phi_{res}}{z} \tag{C-20}$$

Sedimentation is described as a first order process. The resuspension process is not modelled. The resuspension flux should be supplied by the user. It can be entered as a time dependent input variable. The choice of the resuspension flux and the settling velocity governs the level of suspended solids in the water column, which is important

for the distribution of inorganic phosphorus between the dissolved and particulate phases (see eq. C-8)

- **Output Variables**

In order to make direct comparison of the results with frequently monitored variables more convenient, the following output variables are calculated.

$$P_{orto} = P_{anorg} f_{dpano} \tag{C-21}$$

$$P_{tot} = P_{org} + P_{anorg} + Aa_{pc} \tag{C-22}$$

$$N_{kj} = N_{org} + NH_4 + Aa_{nc} \tag{C-23}$$

$$N_{tot} = N_{kj} + NO_3 \tag{C-24}$$

$$Chl - a = a_{chlc} A \tag{C-25}$$

- Parameters

Table C-1 presents a list of all parameters used in EUTROF1. In these table also the default values used and a typical range for some parameters is given.

Symbol	Description	Dimension	Default	Range
a_{chcl}	Chlorophyll to carbon ratio	µg Chl-a/mg C	30	25-250
a_{nc}	Nitrogen to carbon ratio	mg N/mg C	0.25	0.18-0.25
a_{oc}	Oxygen to carbon ratio	mg O_2/mg C	2.67	
a_{pc}	Phosphorus to carbon ratio	mg P/mg C	0.025	0.025-0.050
ε_0	Background extinction	1/m	1.0	1.0-5.0
ε_{alg}	Specific extinction chlorophyll	µg Chl-a/l,m	0.016	0.012-0.025
ε_{SS}	Specific extinction suspended solids	mg SS/l,m	0.050	0.020-0.060
f_{dbod}	Fraction dissolved BOD	-	1	
f_{dnorg}	Fraction dissolved organic nitrogen	-	0	
f_{dporg}	Fraction dissolved organic phosphorus	-	0	
f_{norg}	Fraction algal nitrogen released as NORG	-	1	
f_{porg}	Fraction algal phosphorus released as PORG	-	1	
I_s	Optimal light intensity	W/m^2	40	10-100
k_{BOD}	Oxidation rate constant BOD	1/d	0.1	0.02-3.4
K_{BODO}	Monod constant oxidation BOD	mg O_2/l	2.0	
k_{den}	Denitrification rate constant	1/d	0.1	0.0-0.1
K_{dNO}	Monod constant denitrification	mg O_2/l	0.5	
k_{min}	Mineralisation rate constant	1/d	0.1	0.01-0.40
K_{mN}	Ammonia preference constant	mg N/l	0.025	
K_N	Monod constant nitrogen	mg N/l	0.010	0.01-0.30
k_{nit}	Nitrification rate constant	1/d	0.1	
K_{NO}	Monod constant nitrification	mg O_2/l	2.0	
K_P	Monod constant phosphorus	mg P/l	0.005	0.001-0.05
K_{pip}	Partition constant phosphorus	l/ mg SS	0.010	
k_{res}	Respiration rate constant	1/d	0.1	0.05-0.2
k_{rmin}	Minimum oxygen transfer coefficient	m/d	0.1	
k_{die}	Die rate constant	1/d	0.2	0.0-0.3
θ_{BOD}	Temperature coefficient oxidation BOD	-	1.047	
θ_{den}	Temperature coefficient denitrification	-	1.045	
θ_{ga}	Temperature coefficient algal growth	-	1.047	
θ_{min}	Temperature coefficient mineralisation	-	1.047	
θ_{nit}	Temperature coefficient nitrification	-	1.080	
θ_{rs}	Temperature coefficient respiration	-	1.047	
θ_{rea}	Temperature coefficient reaeration	-	1.024	
μ_{max}	Maximum specific growth rate algae	1/d	4.0	1.0-5.0
v_{so}	Nett settling velocity organic matter	m/d	0.1	0.001-0.1
v_{ss}	Settling velocity suspended solids	m/d	0.1	0.1-5.0

Table C-1 Parameters used in EUTROF1

External variables

Table C-2 presents the external variables used in the model. All input variables mentioned in this table are location specific and time dependent. For some of the variables a typical range is provided. For the surface light intensity and the temperature an example time series is given on the diskette. These series represent actual measured data during a summer month at moderate latitude and can be found in the in the file RIEUT1.EXT.

Symbol	Description	Dimension	Default	Range
SOD	Sediment oxygen demand	g O_2/m^2,d	1.0	0.0-2.0
Pflux	Phosphorus release flux	g P/m^2,d	0.00	0.0-0.01
Nflux	Ammonia release flux	g N/m^2,d	0.00	0.0-0.05
Φres	Resuspension flux suspended solids	g/m^2,d	0.5	0.0-3.0
I_0	Surface light intensity	W/m^2	10.0	
T	Temperature	°C	20.0	

Table C-2 External variables used in EUTROF1

Flow variables

The velocity is calculated from the flow and the cross sectional area, which are defined as flow variables. This means that they are read directly from the hydrodynamic part of duflow.

Symbol	Description	Dimension	Default
Q	Flow	m^3/s	1.0
As	Cross sectional area	m^2	10.0
z	Depth	m	1.0

Table C-3 Flow variables used in EUTROF1

Appendix D EUTROF2

Like the other pre-defined model EUTROF2 is a eutrophication type of model. The main difference between this model and EUTROF1 is the way the sediment water interaction is dealt with. In this model the sediment top layer is modelled to, which enables a dynamic description of the fluxes across the sediment water interface. As the sediment act as the memory of a system with respect to the loading history, this makes the model especially suitable for simulation of longer time scales. EUTROF2 can be used for example to study the effects of reduction in nutrient loads upon the release of nutrients from the sediment.

The water column kinetics are very similar to those used in EUTROF1. Also in this model the cycling of nitrogen, oxygen and phosphorus is modelled. However in EUTROF2 three types of algal species can be defined, which means that also succession and the dynamics of the composition of the algal population can be simulated to a certain extend.

State variables

The following state variables are included in the model:

- A_1,A_2,A_3 Algal Biomass species 1,2 and 3 (mg C/l)
- A_b Total Algal Biomass in the sediment (mg C/l)
- SS_W Suspended Solids concentration (mg/l)
- SS_B Solid concentration in the sediment (mg/l)
- TIP_W Total inorganic phosphorus water column (mg P/l)
- TIP_B Total inorganic phosphorus sediment (mg P/l)
- TOP_W Total organic phosphorus water column (mg P/l)
- TOP_B Total organic phosphorus sediment (mg P/l)
- TON_W Total organic nitrogen (mg N/l)
- TON_B Total organic nitrogen sediment (mg/N/l)
- $NH4_W$ Ammonia nitrogen water column (mg N/l)
- $NH4_B$ Ammonia nitrogen sediment (mg N/l)
- $NO3_W$ Nitrate nitrogen water column (mg N/l)
- $NO3_B$ Nitrate nitrogen sediment (mg N/l)
- $O2_W$ Oxygen water column (mg/l)
- $O2_B$ Oxygen sediment (mg/l)
- BOD_W Biochemical oxygen demand (mg/l)
- BOD_B Biochemical oxygen demand (mg/l)

Sediment model

The degradation of organic matter in the sediment can have an important influence on the concentration of oxygen and nutrients in the overlying water column. Due to the decomposition of organic matter nutrients are released to the interstitial water in the sediment. Furthermore the degradation of organic matter within the sediment results into a flux of oxygen from the overlying water to the sediment (or a flux of oxygen equivalents directed towards the water column in case of anaerobic degradation). This sediment oxygen demand can be a substantial sink for oxygen, while the resulting release of nutrients can be an important contribution to the total nutrient load of a system. Additionally the occurrence of anoxic conditions within the sediment may dramatically increase certain nutrient fluxes. Complex mechanisms of redox reactions and pH control the state and concentration of nutrients and metals and thereby the release of nutrients from the sediment. The relative importance of the sediment water interaction requires the incorporation of a dynamic description of the processes within the sediment and of the transport across the sediment water interface. There are several ways to model the sediment water interaction. In EUTROF2 a relative simple description is used. A general outline of the concept used is given in this paragraph. In the next paragraph a more detailed description of the processes in both the water column and the sediment is presented.

Like in eutrof1 suspended solids are modelled is a simple way. Sedimentation is considered to be a first order process. The resuspension flux should be provided by the user. Eventually the resuspension flux can be related to flow velocity or bottom shears stress. Such relationships however are not included in the process descriptions, because several relationships are available and the user should select the one most appropriate for the water system to be modelled. Sedimentation and resuspension are assumed to occur simultaneously. The following equation describes the suspended solids concentration in the water column:

$$\frac{dSS_W}{dt} = \frac{v_{ss}}{Z} + \frac{F_{res}}{Z} \qquad \text{(D-1)}$$

As the porosity and density of the sediment top layer are considered to be constant and only one fraction suspended solids is taken into account the concentration of sediment is constant and given by:

$$SS_B = \rho * (1 - POR) * 1000 \qquad \text{(D-2)}$$

Because of the sedimentation and resuspension, the sediment water interface is moving with respect to the fixed coordinate system. The velocities by which the benthic surface is displaced can be expressed in terms of the sedimentation and resuspension fluxes:

$$v_s = \frac{F_{sed}}{\rho * (1 - POR) * 1000} \qquad \text{(D-3)}$$

$$v_r = \frac{F_{res}}{\rho * (1 - POR) * 1000} \qquad \text{(D-4)}$$

In which the sedimentation flux F_{sed} is given by:

$$F_{sed} = v_{ss} * SS_W \qquad \text{(D-5)}$$

The net displacement of the interface is given by:

$$v_{sd} = v_s - v_r \qquad \text{(D-6)}$$

Because the depth of the sediment top layer is considered to be constant the interface between the sediment top and lower layer is also moving with a velocity equal to v_{sd}. The equations D-1 to D-6 form the basis of the dynamic description of the sediment water interactions. The concept is illustrated in figure D-1.

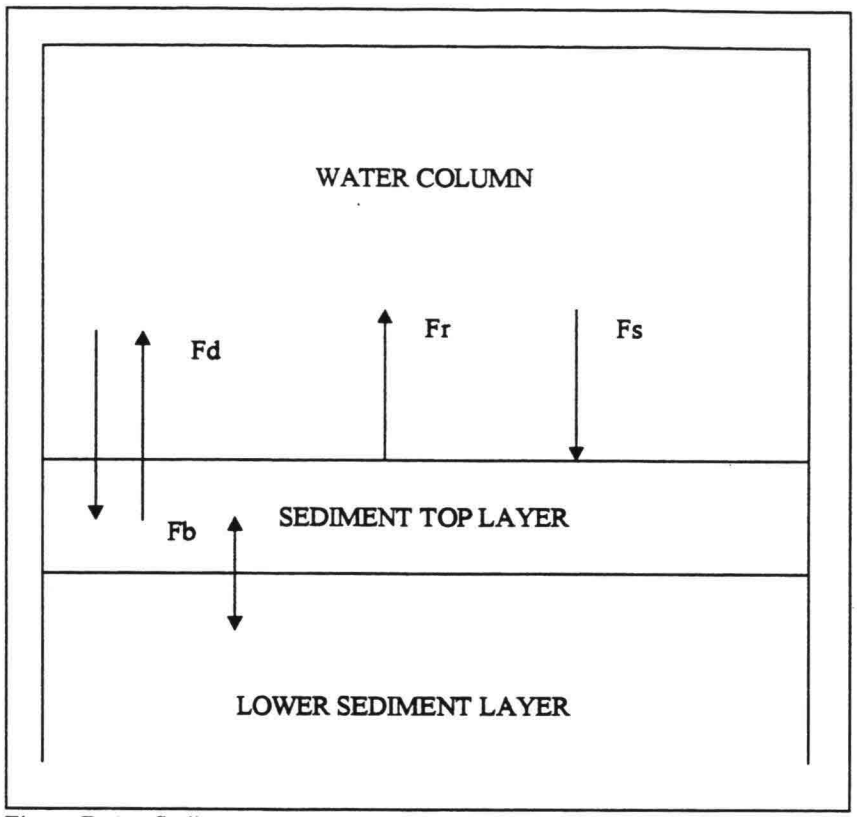


Figure D-1 Sediment model concept

For the description of the exchange fluxes a distinction must be made between dissolved constituents (like ammonia, nitrate and oxygen) and constituent which can be associated with the suspended solids (like inorganic and organic phosphorus, organic nitrogen and bod). These last type of constituents are considered to be present both in a dissolved and particulate form. For a certain constituent X the following forms are distinguished:

$$DX_W = f_{diw} TX_W \qquad \text{(D-7)}$$

$$PX_W = (1 - f_{diw}) \frac{TX_W}{SS_W} \qquad \text{(D-8)}$$

$$DX_B = f_{dib} \frac{TX_W}{POR} \qquad \text{(D-9)}$$

$$PX_B = (1 - f_{dib}) \frac{TX_B}{SS_B} \qquad \text{(D-10)}$$

Where TX_W and TX_B are the total concentrations of constituent X in the water column and the sediment top layer respectively. DX and PX represent the dissolved and particulate fractions. The total sediment concentration is expressed per unit of sediment volume. The dissolved fractions in the water column and sediment are considered to be constant and given by f_{dxw} and f_{dxb}. For inorganic phosphorus these fractions are calculated using linear partition (see eq. D-36 and D-37). The concentration of dissolved constituents in the sediment is expressed per unit of pore water volume and the particulate constituent concentration are given per unit of dry sediment weight in both the water column and sediment top layer.

The total exchange of constituent X across the sediment water interface is represented by the following fluxes:

- **The diffusive exchange flux:**

$$F_{XD} = \frac{E_{diff}}{HB}\left(D_{XB} - D_{XW}\right) \qquad \text{(D-11)}$$

The dissolved fraction is subject to diffusive exchange. The difference between the concentration in the interstitial water and the water column is the driving force for mass transport.

- **The sedimentation flux:**

$$F_{XS} = F_{sed} PX_W + v_s POR\, DX_W \qquad \text{(D-12)}$$

As the particulate fraction is expressed per unit of sediment mass, the flux of constituent X across the interface is equal to the sedimentation flux of suspended solids multiplied with the particulate constituent concentration. The second term in eq. D-12 describes the inclusion of pore water due to the formation of new sediment by sedimentation.

- **The resuspension flux:**

$$F_{XR} = F_{res} PX_B + v_r POR\, DX_B \qquad \text{(D-13)}$$

The resuspension of particulate X is given by the product of the resuspension flux of solids and the particulate concentration in the sediment. The second term of eq. D-13 represents the release of pore water during resuspension.

- **Transport between top and lower sediment layer:**

$$F_{XB} = -v_{sd} TX_B \qquad \text{if } v_{sd} < 0 \qquad \text{(D-14)}$$

or:

$$F_{XB} = -v_{sd} TX_{LB} \qquad \text{if } v_{sd} < 0 \qquad \text{(D-15)}$$

Because of the concept of a constant top layer depth there is a transport of sediment between the top and lower sediment layer if the net displacement velocity v_{sd} is not equal to 0. If net sedimentation occurs sediment is transported from the top toward the lower layer. In case of net resuspension the sediment top layer is replenished with sediment from the lower layer. The concentration in the lower sediment layer is considered to be constant and should be supplied by the user. Diffusive exchange between the two sediment layers is not taken into account. Hence the concentration in

the top layer is only influenced by the quality of the lower layer if net resuspension occurs.
The total transport across the interface is equal to the sum of the fluxes expressed above. The equations describing the concentration in the water column and the sediment top layer are given by:

$$\frac{dX_W}{dt} = \frac{F_{XD} - F_{XS} + F_{XR}}{Z} + P_{XW} \qquad \text{(D-16)}$$

and:

$$\frac{dX_B}{dt} = \frac{F_{XD} - F_{XS} + F_{XR} + F_{XB}}{HB} + P_{XB} \qquad \text{(D-17)}$$

Equations D-16 and D-17 also can be used for constituents only present in a dissolved form. For these constituents the individual fluxes can be expressed like:

$$F_{XD} = \frac{E_{diff}}{HB}\left(X_I - X_W\right) \qquad \text{(D-18)}$$

$$F_{XS} = v_s POR\, X_W \qquad \text{(D-19)}$$

$$F_{XR} = v_r POR\, X_I \qquad \text{(D-20)}$$

$$F_{XB} = -v_{sd} X_B \qquad if\ v_{sd} > 0 \qquad \text{(D-21)}$$

or

$$F_{XB} = v_{sd} X_{LB} \qquad if\ v_{sd} < 0 \qquad \text{(D-22)}$$

The concentration in the interstitial water in this case is represented by X_I and equal to X_B/POR. For dissolved constituents the concentration in the interstitial is also influenced by sedimentation and resuspension, because of the inclusion or release of pore water respectively.

Process descriptions

In the following part the equations, describing the processes are presented. For each state variable the processes in both the sediment and the water column are given. The transport fluxes across the water sediment interface already have been described in eq. D-16 and D-17 and are not included in the equations below. The explanation of the symbols used in this part is given in the tables at the end of this appendix.

- **Algae**

In this model three algae species can be simulated. So the succession and dynamics of the composition of the algae population can be simulated to a certain extend.

The overall growth equation for each species is given by:

$$\frac{dA_{W,i}}{dt} = [\mu_{max,i} F_{T,i} F_{N,i} F_{L,i}] A_{W,i} - [K_{die,i} + k_{res,i} \theta_{ra,i}^{(T-20)} - \frac{v_{sa,i}}{Z}] A_{w,i} \qquad \text{(D-23)}$$

The growth is considered to be limited by nutrients, light and temperature.

Nutrient limitation is described as:

$$F_{Ni} = \min\left[\frac{DIP_W}{DIP_W + k_P i}, \frac{DIN_W}{DIN_W + k_{ni}}\right] \qquad \text{(D-24)}$$

Where DIN_W is the total inorganic nitrogen concentration equal to the sum of nitrate and ammonia in the water column. The reduction of the maximum growth rate is controlled by the most limiting factor. It is assumed that algae can use both ammonia and nitrate for their growth. The uptake of both nitrogen constituents is controlled by the ammonia preference factor (see eq. D-40).

Because EUTROF2 is intended for simulation of long time scales a daily averaged light limitation function is used. The depth integrate Steele equation (see Appendix C, eq. C-3) is integrated over the daylight period. This means that EUTROF2 is not able to describe diurnal variations in algal growth. The light limitation factor is expressed as:

$$F_{Li} = \frac{ef}{\varepsilon_{tot} Z}\left[\exp(-\alpha_{1i}) - \exp(-\alpha_{0i})\right] \qquad \text{(D-25)}$$

in which:

$$\alpha_{0i} = \frac{I_a}{I_{si}} \qquad \text{(D-26)}$$

and:

$$\alpha_{1i} = \alpha_{0i} \exp(-\varepsilon_{tot} Z) \qquad \text{(D-27)}$$

I_a is the average light intensity during the daylight period (L) and f is the fraction of daylight during the day (equal to L/24).
The total extinction coefficient is determined by the background extinction of the water and the contributions of chlorophyll and suspended solids to the vertical light attenuation.

$$\varepsilon_{tot} = \varepsilon_0 + \varepsilon_{alg} Chl - a + \varepsilon_{SS} SS \qquad \text{(D-28)}$$

For internal computational purpose algal carbon is used as a measure for the biomass. The algal C concentration is converted to chlorophyll-a using a fixed chlorophyll to carbon ratio for each species. The total chlorophyll concentration can be expressed as:

$$Chl-a=\sum_{i=1}^{3}a_{ChlaC,i}A_{Wi} \tag{D-29}$$

Temperature dependency of algal growth is described in a different way as in EUTROF1. For the individual species an optimum curve is used to simulated temperature dependent growth. The temperature limitation factor is given by:

$$F_{Ti}=\frac{T_{csi}-T}{T_{csi}-T_{osi}}\exp\left(1-\frac{T_{csi}-T}{T_{csi}-T_{osi}}\right) \tag{D-30}$$

If the water temperature is above the critical temperature for growth

$F_{Ti}=0$.

Three loss processes are included in the algal balance equation (eq. D-23). The endogenous respiration is considered to be temperature dependent. The second loss term represents the die-off and the effects of grazing and is regarded to be constant. Finally the sedimentation of algae is included. Although the sedimentation velocity of the algae is low, the total load settling to the sediment can be substantial. Together with the sedimentation of dead organic matter (detritus and from man made sources) it determines the organic and nutrient load of the sediment and controls the resulting interaction between the sediment and the overlying water column. Once settled into the sediment the algae are converted to benthic organic carbon and subject to anaerobic decomposition. There is no transport of living algae from the sediment to the water column. As the stoichiometric ratio for all algae species are considered to be the same for the benthic algal carbon concentration only one state variable has to be defined. The following equation is used to describe the algae concentration in the sediment:

$$\frac{dA_B}{dt}=-K_{daB}\theta_{daB}^{(T-20)}A_B \tag{D-31}$$

- **Organic Phosphorus**

During respiration and die-off of the algae, part the associated phosphorus is released as organic phosphorus, the remaining part is distributed to the inorganic phosphorus pool. The phosphorus to carbon ration is assumed to be constant and the same for all three algae species. Due to aerobic mineralisation in the water column organic phosphorus is converted to the inorganic form. Organic phosphorus is both present in a dissolved and particulate form. The dissolved fraction is assumed to be constant and equal to f_{dpoW}. The following equation is used to describe the total organic phosphorus concentration in the water column:

$$\frac{dTOP_W}{dt}=-K_{\min}\theta_{\min}^{(T-20)}TOP_W+f_{porg}a_{pc}\sum_{i=1}^{3}\left[(K_{die,i}+K_{res,i}\theta_{ra,i})A_{wi}\right] \tag{D-32}$$

In the sediment organic phosphorus is only subject to anaerobic decomposition. The total organic phosphorus in the sediment top layer is given by:

$$\frac{dTOP_B}{dt}=-K_{\min B}\theta_{\min B}^{(T-20)}TOP_B+a_{pc}K_{daB}\theta_{daB}^{(T-20)}A_B \tag{D-33}$$

- **Inorganic Phosphorus**
The equations describing the inorganic phosphorus concentration in the water column and the sediment top layer are given by:

$$\frac{dTIP_W}{dt} = K_{\min}\theta_{\min}^{(T-20)}TOP_W - a_{pc}\sum_{i=1}^{3}\left[\mu_{\max i}F_{Ti}F_{Ni}F_{Li}A_{Wi}\right] + (1-f_{porg})a_{pc}\sum_{i=1}^{3}\left[(K_{diei}+K_{resi}\theta_{rai}^{(T-20)})A_{Wi}\right] \quad \text{(D-34)}$$

and:

$$\frac{dTIP_B}{dt} = K_{\min B}\theta_{\min B}^{(T-20)}TOP_B \quad \text{(D-35)}$$

Inorganic phosphorus is formed during aerobic and anaerobic mineralisation in the water column and sediment respectively. It is also released during the algal respiration and die-off. Part of the inorganic phosphorus is adsorbed to the suspended solids. The dissolved fractions in the water column and in the interstitial water are calculated, using linear partition:

$$f_{dpW} = \frac{1}{1+K_{pipW}SS_W} \quad \text{(D-36)}$$

$$f_{dpB} = \frac{1}{1+K_{pipB}SS_B} \quad \text{(D-37)}$$

The use of equations D-36 and D-37 implies that is assumed that the equilibrium is reached instantaneously. The sorption rate is considered to be fast compared to the other relevant processes in the phosphorus cycle. Furthermore it is assumed that the linear part of the sorption isotherm may be used.

- **Organic Nitrogen**
The behaviour of organic nitrogen is similar to that of organic phosphorus. In the water column release during algal loss processes and anaerobic mineralisation takes place. In the sediment the anaerobic mineralisation of settled algae and organic nitrogen are the controlling processes. The total organic nitrogen concentration in the water column and sediment top layer are given by:

$$\frac{dTON_W}{dt} = -K_{\min}\theta_{\min}^{(T-20)}TON_W + f_{norg}a_{nc}\sum_{i=1}^{3}\left[(K_{die,i}+K_{res,i}\theta_{ra,i})A_{Wi}\right] \quad \text{(D-38)}$$

$$\frac{dTON_B}{dt} = -K_{\min B}\theta_{\min B}^{(T-20)}TON_B + a_{nc}K_{daB}\theta_{daB}^{(T-20)}A_B \quad \text{(D-39)}$$

- **Ammonia Nitrogen**
During algal respiration and die-off of the algae part of the nitrogen is released as ammonia. The remaining part is added to the pool of organic nitrogen. Both ammonia and nitrate can be used for algal growth. The preference for the nitrogen source used is controlled by the nitrogen preference factor given by:

$$P_{NH4} = NH4_W \frac{NO3_W}{(K_{mN} + NH4_W)(K_{mN} + NO3_W)} + NH4_W \frac{K_{mN}}{(NH4_W + NO3_W)(K_{mN} + NO3_W)} \quad \text{(D-40)}$$

The nitrification rate in the water column is controlled by the oxygen concentration, using a Monod type of equation. Depending on the value of K_{NO} the rate can be limited at low oxygen concentrations. The equation for ammonia nitrogen in the water column is given by:

$$\frac{dNH4_W}{dt} = -K_{nit}\theta_{nit}^{(T-20)} \frac{O2_W}{(O2_W + K_{no})} NH4_W + K_{\min}\theta_{\min}^{(T-20)} TON_W - a_{nc} P_{NH4} \sum_{i=1}^{3} [\mu_{\max i} F_{Ti} F_{Ni} F_{Li} A_{Wi}] + a_{nc} \sum_{i=1}^{3} [K_{diei} + k_{resi}\theta_{rai}^{(T-20)}] \quad \text{(D-41)}$$

Organic nitrogen is hydrolysed to ammonia by bacterial action within the sediment. As the decomposition processes in the bottom are considered to be anaerobic, it is assumed that there is no oxygen in the sediment top layer. Hence no nitrification occurs in the sediment. The equation describing the sediment ammonia concentration is given by:

$$\frac{dNH4_B}{dt} = K_{\min}\theta_{\min}^{(T-20)} TON_B \quad \text{(D-42)}$$

- **Nitrate Nitrogen**

In the water column nitrate is formed during nitrification. Depending on the ammonia preference factor nitrate can be used as a nitrogen source for algal growth. Denitrification, which is also controlled by the oxygen concentration is included too. The nitrate concentration in the water column is given by:

$$\frac{dNO3_W}{dt} = -K_{den}\theta_{den}^{(T-20)} \frac{K_{dno}}{(K_{dno} + O2_W)} NO3_W + K_{nit}\theta_{nit}^{(T-20)} \frac{O2_W}{(O2_W + K_{no})} NH4_W - a_{nc}(1 - P_{NH4}) \sum_{i=1}^{3} [\mu_{\max i} F_{Ti} F_{Ni} F_{Li} A_{Wi}] \quad \text{(D-43)}$$

In the bottom the only process is denitrification. Nitrate is present in the sediment due to the diffusive transport from the overlying water column. The nitrate concentration in the sediment top layer is given by:

$$\frac{dNO3_B}{dt} = -K_{denB}\theta_{denB}^{(T-20)} NO3_B \quad \text{(D-44)}$$

- **Cbod**

Because in practice 5 day carbon bod values are used, the equation describing the BOD concentration are expressed in BOD_5. In the oxygen balance equation (eq. D-48) however the ultimate BOD is used. A conversion factor is used to calculate BOD_5 from BOD_u. The BOD, which is produced by die-off of the algae and the BOD used as a carbon source during denitrification is corrected using this conversion factor. The conversion factor is given by:

$$X_{conv} = 1 - \exp(-5K_{bod}^{20}) \quad \text{(D-45)}$$

The oxidation of BOD in the water column is temperature dependent and limited at low oxygen concentration by a Monod type of kinetic.
Water column BOD_5 is given by:

$$\frac{dBOD_W}{dt} = -K_{bod}\theta_{bod}^{(T-20)}\frac{O2_W}{(O2_W + K_{bodo})}BOD_W + \left[a_{oc}\sum_{i=1}^{3}\left[K_{die,i}A_{Wi}\right] - \frac{5}{4}\frac{32}{14}K_{den}\theta_{den}^{(T-20)}\frac{K_{dno}}{(K_{dno} + O2_W)}NO3_W\right]X_{conv} \quad \text{(D-46)}$$

In the sediment the settled algae and benthic organic matter are subject to anaerobic degradation. In reality the reaction mechanisms involved are very complex. In the model only the initial step in which the organic carbon is converted to reactive intermediates is included. This formulation is similar and consistent with the degradation of organic nitrogen and phosphorus within the sediment. The reactive intermediates however participate in further reactions. For example volatile acids react to methane. In the model the redox reactions oxidizing these intermediates are not included, but these reduced carbon products are expressed as negative oxygen equivalents that are transported across the sediment water interface. This concept first introduced by Di Toro and Connolly enables a dynamic description of the sediment oxygen demand. A further explanation of the concept is given in the part on oxygen below. The equation describing organic carbon expressed as BOD_5 is given by:

$$\frac{dBOD_B}{dt} = \frac{a_{oc}K_{daB}\theta_{daB}^{(T-20)}A_B - \frac{5}{4}\frac{32}{14}K_{denitB}\theta_{denitB}^{(T-20)}NO3_B}{X_{conv}} - K_{bodB}\theta_{bodB}^{(T-20)}BOD_B \quad \text{(D-47)}$$

- **Oxygen**

Dissolved oxygen in the water column is described by:

$$\frac{dO2_W}{dt} = K_{re}\theta_{re}^{(T-20)}(C_s - O2_W) - K_{BOD}\theta_{BOD}^{(T-20)}\frac{O2_W}{(O2_W + K_{bodo})}\frac{BOD_W}{X_{conv}} - \frac{64}{14}K_{nit}\theta_{nit}^{(T-20)}\frac{O2_W}{(O2_W K_{NO})}NH4_W - \frac{32}{12}\sum_{i=1}^{3}\left[K_{resi}\theta_{resi}^{(T-20)}A_{Wi}\right] \sum_{i=1}^{3}\left[\mu_{maxi}F_{Ti}F_{Ni}F_{Li}A_{Wi}\right]\frac{32}{12} + \frac{48}{12}a_{nc}(1 - P_{nh4})NO3_W \quad \text{(D-48)}$$

Additional to the oxidation of carbon BOD algal respiration and nitrification are included as oxygen consuming processes.
Reaeration is described using a empirical equation for the mass transfer coefficient. This coefficient is related to the flow velocity and water depth using:

$$k_{mas} = 3.94u^{0.5}z^{-1.5} \quad \text{(D-49)}$$

or if $k_{mas} < k_{r\min}$

$$k_{mas} = k_{r\min} \quad \text{(D-50)}$$

The dimension of the velocity is m/s and the resulting k_{mas} is in m/day. At low stream velocity the use of eq. D-49 can result into extremely low values for the mass transfer

coefficient. The user can define a minimum value for k_{mas}, which is used as a lower bound for the mass tranfer coefficient. The reaeration constant k_{re} is given by:

$$k_{re} = \frac{k_{mas}}{Z} \qquad \text{(D-51)}$$

Production of oxygen results from primary production. In case nitrate is used as a source for nitrogen an additional oxygen production takes place, because of the reduction of nitrate during the assimilation process.

The following equation is used to describe the sediment `oxygen concentration':

$$\frac{dO2_B}{dt} = -K_{bodB}\theta_{bodB}^{(T-20)}\frac{BOD_B}{X_{conv}} \qquad \text{(D-52)}$$

As stated above in the part on BOD, the organic carbon and the settled algae are mineralised anaerobically and expressed as BOD. Both reactions are considered to be sinks for oxygen and quickly drive the oxygen concentration within the sediment top layer negative. This negative oxygen concentration indicates that the redox state in the sediment is rather reduced than oxidized. The calculated negative concentration is considered to be oxygen equivalence of the reduced intermediate products produced in the mineralisation reaction mechanism. It is assumed that the reduced carbon intermediates (expressed as oxygen equivalents) are transported across the sediment interface and are oxidized to CO_2 en H_2O in the overlying water column. The sediment oxygen demand is in fact calculated as the transport of oxygen and oxygen equivalents across the sediment water interface and is controlled by the decomposition of organic carbon in the sediment and the overlying water dissolved oxygen concentration. The SOD in this concept is described by:

$$SOD = \frac{E_{diff}}{HB}(O2_W - O2_B) \qquad \text{(D-53)}$$

Parameters

Table D-1 presents a list of all parameters used in EUTROF2. In this table also the default values used are given.

Symbol	Description	Dimension	Default
$a_{chl,i}$	Chlorophyll to carbon ratio species i	µg Chl -a/mg C	30.000
a_{ac}	Nitrogen to carbon ratio	mg N/mg C	0.25
a_{oc}	Oxygen to carbon ratio	mg O_2/m g C	2.670
a_{pc}	Phosphorus to carbon ratio	mg P/mg C	0.025
BOD_{LB}	BOD concentration lower sediment layer	mg O_2/l	3500.0
ε_0	Background extinction	1/m	1.0
ε_{alg}	Specific extinction chlorophyll	µg Chl -1/l,m	0.016
E_{dif}	Diffusive exchange rate constant	m^2/d	0.0002
ε_{SS}	Specific extinction suspended solids	mg SS/l, m	0.05
f_{dBODB}	Fraction dissolved BOD sediment	-	0.03
f_{dBODW}	Fraction dissolved BOD water column	-	0.90
f_{dNOB}	Fraction dissolved organic nitrogen sediment	-	0.10
f_{dNOB}	Fraction dissolved organic nitrogen water column	-	0.10
f_{dPOB}	Fraction dissolved organic phosphorus sediment	-	0.10
f_{dPOW}	Fraction dissolved organic phosphorus water column	-	0.10
f_{norg}	Fraction algal nitrogen released as organic nitrogen	-	0.50
f_{porg}	Fraction algal phosphorus released as organic phosphorus	-	0.50
HB	Depth of the sediment top layer	m	0.10
$I_{s,i}$	Optimal light intensity species 1	W/m^2	40.0
K_{bod}	Oxidation rate constant BOD water column	1/d	0.10
K_{bodB}	Anaerobic decomposition rate constant BOD sediment	1/d	0.005
K_{bodo}	Oxygen half saturation constant BOD decay	mg O_2/l	2.0
K_{daB}	Anaerobic decay rate constant algae sediment	1/d	0.010
K_{den}	Denitrification rate constant water column	1/d	0.10
K_{denB}	Denitrification rate constant sediment	1/d	0.10
$K_{dei,i}$	Die-off rate constant species i	1/d	0.10
K_{dno}	Oxygen half saturation constant denitrification	mg O_2/l	0.50
K_{min}	Decomposition rate constant organic matter water column	1/d	0.10
K_{minB}	Anaerobic decomposition rate constant sediment	1/d	0.001
K_{nn}	Ammonia preference constant	mg N/l	0.025
$k_{n,i}$	Monod constant nitrogen algal growth species i	mg N/l	0.025
K_{nit}	Nitrification rate constant	1/d	0.10
K_{no}	Oxygen half saturation constant nitrification	mg O_2/l	2.0
$k_{p,i}$	Monod constant phosphorus algal growth species i	mg P/l	0.005
K_{pipB}	Partition coefficient inorganic phosphorus sediment	l/mg SS	0.00250
K_{pipW}	Partition coefficient inorganic phosphorus water column	l/mg SS	0.05000
$k_{res,i}$	Respiration rate constant species i	1/d	0.05000
k_{min}	Minimum oxygen transfer coefficient	m/d	0.40000
$NH4_{LB}$	Ammonia concentration lower sediment layer	mg N/l	1.600
$NO3_{LB}$	Nitrate concentration lower sediment layer	mg N/l	0.000
$O2_{LB}$	Oxygen (equivalents)concentration lower sediment	mg O_2/l	0.000
POR	Sediment porosity	-	0.8
ρ	Density suspended solids	kg/m^3	1320
θ_{bod}	Temperature coefficient oxidation BOD	-	1.04
θ_{bodB}	Temperature coefficient decomposition BOD sediment	-	1.08
$T_{ce,i}$	Critical temperature species i	°C	35.0
θ_{daB}	Temperature coefficient anaerobic decomposition sediment	-	1.080
θ_{den}	Temperature coefficient denitrification water column	-	1.040
θ_{denB}	Temperature coefficient denitrification sediment	-	1.040
TIP_{LB}	Total inorganic phosphorus lower sediment layer	mg P/l	1.900
θ_{min}	Temperature coefficient mineralisation water column	-	1.040
θ_{minB}	Temperature coefficient mineralisation sediment	-	1.080
θ_{nit}	Temperature coefficient nitrification	-	1.080
TON_{LB}	Total organic nitrogen lower sediment layer	mg N/l	15.000
TOP_{LB}	Total organic phosphorus lower sediment layer	mg P/l	5.000
$T_{oc,i}$	Optimal temperature species i	°C	20.0
$\theta_{re,i}$	Temperature coefficient respiration species i	-	1.040
θ_{rea}	Temperature coefficient reaeration	-	1.024
$\mu_{max,i}$	Maximum growth rate species i	1/d	1.200
$v_{sa,i}$	Settling velocity species i	m/d	0.00500
v_{SS}	Settling velocity suspended solids	m/d	0.000

Table D-1 Parameters used in EUTROF2

External variables

Table D-2 presents the external variables in the model. For some of the variables a typical value is provided. For the daily averaged light intensity, day length and temperature a time series of one year is available on diskette in the file RIEUT2.EXT. Light intensity and water temperature are measured at moderate latitude.

Symbol	Description	Dimension	Default Range
F_{res}	Resuspension flux	g/m^2, d	5.00
T	Temperature	°C	20.0
I_a	Average light intensity	W/m^2	100.0
L	Day length	hour	12.0

Table D-2 External variables used in EUTROF2

Flow variables

Symbol	Description	Dimension	Default	Range
Z	Water depth	m	1.0	
As	Cross sectional area	m^2	20.0	
Q	Flow	m^3/s	1.0	

Table D-3 Flow variables used in EUTROF2

The velocity is calculated from the flow and the cross sectional area, which are defined as flow variables. This means that they are read directly from the hydrodynamic part of DUFLOW.

Initial conditions of some sediment variables

Table D-4 presents some typical values for the concentration is the sediment. These values can be very site specific and the model is rather sensitive to the initial sediment concentrations. If no data are available table D-4 provides some guide lines, which can be used as an initial guess.

Symbol	Description		Dimension	Range
NH_4I	Interstitial ammonia concentration		mg N/l	0.2-5.0
$NO3_I$	Interstitial nitrate concentration		mg N/l	0.0-0.1
DIP_B	Interstitial inorganic phosphorus		mg P/l	0.0-0.1
PIP_B	Particulate inorganic phosphorus		mg P/g SS	0.0-3.0
TON_B	Total organic nitrogen	0.5-5 % of the organic matter		
TOP_B	Total organic phosphorus	0.2-1 % of the organic matter		
$PBOD_B$	Particulate BOD	1.4 * organic matter content organic matter content 0-50 % of the dry weight		
$DBOD_B$	Dissolved BOD interstitial water		mg O_2/l	100.0

Table D-4 Some typical sediment characteristics

Appendix E File descriptions

Project file

The file is shown below, the case described in this appendix is EUTROF1 (see appendix C).

```
RIEUT1. CTR
RIVER.  NET
RIVER.  NOD
RIVER.  BEG
RIVER.  BND
RIVER.  RES
EUTROF1.MOB
RIEUT1. BEK
RIEUT1. BNK
RIEUT1. EXT
RIEUT1. PRM
RIEUT1. REK
```

The file contains a list of the files needed to run the complete model. When entering "FILE NAMES SPECIFICATION" (see paragraph 3.7) the files can be supplied for the specific project.

As an extra, an example of a measured data file (MEASUR.MSR) is added to this appendix. The name of this file is not stored in the PROJECT FILE.

In general the positions in the files are in use as following.

- For integers the program reserves 4 positions.
- For comments the program reserves 4 positions.
- For a real format the program reserves 6 positions, including the decimal. If the real format needs more then 6 positions, the format is written as a scientific format.

All the values are separated by one space position.

Control file

The concerned file for the project (appendix C) is shown below. The file contains the input data entered in "CONTROL DATA" (see paragraph 3.4.1).

```
+APPENDIX-C
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIEUT1.CTR
* Control data      program version: 2.xx
*
TIME 910701[a]    0[b] 910731[c]    0[d] 910701[e]    0[f]
CONT 30.000[g] 240.00[h]          0[i]    0[j]    0[k]
       0.00[l]  0.45[m]           0[n]    0[o]    0[p]
     1.0000[q]    1[r] 60.000[s]  0.55[t]    1[u]    1[v]
     PVAK          *
OUTS A      [w]
OUTV a      [x]
OUTV bod    [y]
OUTV nh4    [z]
OUTV no3    [aa]
OUTV norg   [ab]
OUTV o2     [ac]
OUTV panorg [ad]
OUTV porg   [ae]
OUTV chla   [af]
```

The following list gives an explanation about the items in the file.

If the first line starts with: +, this indicates which run is entered in "identification text" (see paragraph 3.4.1.1). The lines 2 till 4 gives information about the related project as comment.

a "Start of computation" (date) in yymmdd
b "Start of computation" (time) in hhmm
c "End of computation" (date) in yymmdd
d "End of computation" (time) in hhmm
e "Start of output" (date) in yymmdd
f "Start of output" (time) in hhmm
g "Time step flow" in minutes
h "Time step output" in minutes
i "Echo boundary" 0=no 1=yes
j "Echo network data" 0=no 1=yes
k "Optimize network" 0=no 1=yes
l "Resistance formula" 0=Manning 1=De Chezy.
m meaningless
n 1 - "theta", hydraulic part of the model.
o "Calc. of advection term" 0=Total 1=Neglected 2=Damped.
p "Extra iteration" 0=No 1=Yes
q "Alpha (corr. for velocity distribution)"
r "Minimum # timesteps between triggers"
s "Timestep quality" in minutes
t "Theta", quality part of the model.
u "Decouple" 0=No 1=Yes

v "Create intermediate flow file" 0=No 1=Yes

* Dependent on the specification in "Locations for output", several lines will be inserted representing that particular specification. For instance: "Extended"=All;

```
PVAK    1       2       3       4       5       6
PVAK    7       8       ^
```

w "Locations for output", section number(s) or structure number(s)(+300) separated by a comma or a dash, or All.

x "Quality variables for output", selected variable 1=a.

y "Quality variables for output", selected variabel.2=bod

z "Quality variables for output", selected variabel.3=nh4

aa "Quality variables for output", selected variabel.4=no3

ab "Quality variables for output", selected variabel.5=norq

ac "Quality variables for output", selected variabel.6=o2

ad "Quality variables for output", selected variabel.7=panorg

ae "Quality variables for output", selected variabel.8=porg

af "Quality variables for output", selected variabel.9=chla

Network file

This file contains entered data from:

- "NETWORK DEFINITION" (paragraph 3.4.2.1.1) (a-d)
- "SECTIONS" (see paragraph 3.4.2.1.3) (e-k)
- "CROSS-SECTIONS" (see paragraph 3.4.2.1.4) (l-o)
- "STRUCTURES" (see paragraph 3.4.2.1.5)

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIVER.NET
*   Network data       program version: 2.xx
*
SECT   1[a]      1[b]     1[c]     2[d]  5000[e] -2.00[f] -2.00[g] 30.00[h] 30.00[i]
W              270.0[j]   3.0[k]
H     0.0000[l]2.0000[m]
BS    10.000[n]10.000[o]

SECT   2         2        2        3     5000    -2.00    -2.00    30.00    .30.00
W              270.0      3.0
H     0.0000 2.0000 2.5000 3.0000
BS1   10.000[p]10.000 11.000 14.000
BS2   10.000[q]10.000 13.000 15.000
BB1   10.000[r]10.000 13.000 20.000
BB2   10.000[s]10.000 13.000 22.000

SECT   3         3        3        4     5000    -2.00    -2.00    30.00    30.00
W              270.0      3.0
H     0.0000 2.0000
BS    10.000 10.000
A1             18.000[t]
A2             18.010[u]
R              1.3000[v]
C+             29.000[w]
C-             31.000[x]

SECT   4         4        4        5     5000    -2.00    -2.00    30.00    30.00
W              270.0      3.0
H     0.0000 2.0000
BS    10.000 10.000
```

The following list gives explanation about the items in the file.

a "section/structure" name by the user.
b section number (+300).
c "Begin Node" of the section.
d "End Node" of the section.
e "Length" of the section.
f "Bottom level", begin.
g "Bottom level", end.
h "Resistance", positive direction.
i "Resistance", negative direction.

j "Direction"
k "Windconv"
l 1st "Depth to bottom"
m 2nd "Depth to bottom", etc.
n 1st "Flow width".
o 2nd "Flow width", etc.
p 1st "Flow width" at the begin of the section.
q 1st "Flow width", at the end of the section.
r 1st "Storage width" at the begin of the section.
s 1st "Storage width", at the end of the section.
t Adjusted "Flow area" at the begin of the section.
u Adjusted "Flow area" at the end of the section.
v Adjusted "Hydraulic radius"
w Adjusted "Resistance" in the positive direction.
x Adjusted "Resistance" in the negative direction.

This is an example where no structures are defined. The next part will deal with the available structures in DUFLOW. See "STRUCTURES" in paragraph 3.4.2.1.5. The following structures are involved:

- Overflow;
- Underflow;
- Siphon;
- Rectangular culvert;
- Elliptic culvert;
- Pump;

- **Overflow.**

Below is shown an overflow example.

```
STRU   1[a]    301[b]    2[c]      3[d]     0[e] -1.00[f] -5.00[g] 999.0[h]
MU   0.990[i] 0.980[j]
```

The following list gives explanation about the items in an overflow.

a "section/structure" name by the user (no meaning).
b structure number (+300).
c "Begin Node" of the overflow.
d "End Node" of the overflow.
e "Length" of the overflow.
f "Sill level", overflow.
g "Width", overflow.
h "Gate level", in case off an overflow is set to 999.0.
i "Mu", positive direction overflow.
j "Mu", negative direction overflow.

- **Underflow**

The next example deals with an underflow.

```
STRU   2[a]    302[b]    4[c]    5[d]    0[e]  -1.75[f]  9.99[g]  -0.25[h]
MU   1.0E-8[i] 0.970[j] 0.960[k] 0.800[l]
```

The following list gives explanation about the items in an underflow.

a "section/structure" name by the user.
b structure number (+300).
c "Begin Node" of the underflow.
d "End Node" of the underflow.
e "Length" of the underflow.
f "Sill level", underflow.
g "Width", underflow.
h "Gate level", underflow.
i "Mu free surface", in positive direction.
j "Mu free surface", in negative direction.
k "Mu submerged flow", in positive direction.
l "Mu submerged flow", in negative direction.

- **Rectangular culvert**

The next example deals with a culvert.

```
STRU   3[a]    303[b]    6[c]    7[d]   100[e]  -1.80[f]  8.00[g]  1.40[h]
MU   0.61[i]  0.62[j]  0.99[k]  0.83[l]  2.5[m]
```

The following list gives explanation about the items in an culvert.

a "section/structure" name by the user.
b structure number (+300).
c "Begin Node" of the culvert.
d "End Node" of the culvert.
e "Length" of the culvert.
f "Sill level", culvert.
g "Width", culvert.
h "Gate level", culvert.
i "Mu free surface", in positive direction.
j "Mu free surface", in negative direction.
k "Mu submerged flow", in positive direction.
l "Mu submerged flow", in negative direction.
m "Chezy coefficient", culvert.

- **Elliptic culvert**

The next example deals with a culvert.

```
STRU   3[a]    303[b]    6[c]    7[d]   100[e]  3.80[f]  2.50[g]  1.40[h]
ELLI  1.60[i]  0.99[j]  0.99[k]  0.83[l]  50.0[m]
```

The following list gives explanation about the items in an culvert.

a "section/structure" name by the user.
b structure number (+300).
c "Begin Node" of the culvert.
d "End Node" of the culvert.
e "Length" of the culvert.
f "Horizontal radius", culvert.
g "Vertical radius", culvert.
h "Inside level", culvert.
i "Sill level", culvert.
j "Mu free surface", in positive direction.
k "Mu free surface", in negative direction.
l "Mu submerged flow".
m "Chezy coefficient", culvert.

- **Siphon**

The next example deals with a siphon.

```
STRU   4[a]    304[b]    8[c]    9[d]   1000[e]  1.10[f]  0.68[g] 10.00[h]
0.780[i]
SIPH   0.00[j] -0.50[k] -1.50[l] -1.25[m]  0.00[n]
```

The following list gives explanation about the items in a siphon.

a "section/structure" name by the user.
b structure number (+300).
c "Begin Node" of the siphon.
d "End Node" of the siphon.
e "Length" of the siphon.
f "Diameter", of the siphon.
g "Mu", in positive direction at the begin of the siphon.
h "Mu", in negative direction at the begin of the siphon
i "Mu", in positive direction at the end of the siphon.
j "Mu", in negative direction at the end of the siphon.
k "Start level", at the begin of the siphon.
l "Stop level", at the begin of the siphon.
m "Start level", at the end of the siphon.
n "Stop level", at the end of the siphon.

- **Pump**

The next example deals with a pump.

```
STRU   5[a]    305[b]   15[c]   16[d]
PUMP 0.6000[e]  0.93[f]  0.10[g]
```

The following list gives explanation about the items in a pump.

a "section/structure" number by the user.

b structure number by duflow.
c "Begin Node" of the pump.
d "End Node" of the pump.
e "Length" of the siphon.
f "Capacity", of the pump.
g "Start level beg.".
h "Stop level beg.".

Nodes file

In this file all data related to the nodes are stored.

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIVER.NOD
*   Network data       program version: 2.xx
*
* file :C:\DUFLOW\RIEUT1\RIVER.NOD
*  2.00
*
+FI   a    0.0b
    1c     5000d        0e     0E+00f   0.00g
    2      10000        0      0E+00    0.00
    3      15000        0      0E+00    0.00
    4      20000        0      0E+00    0.00
    5      25000        0      0E+00    0.00
```

The following list gives explanation about the items in the file.

a Code.
b Orientation of network.
c "Node" number.
d "X-Coordinate" of the 1st node.
e "Y-Coordinate" of the 1nd node.
f "Catchment area" of the 1st node.
g "Runoff factor" of the 1st node.

Initial conditions - Flow file

In this file all initial conditions related to the flow are stored.

Input is entered by "initial conditions" (see paragraph 3.4.2.2, 3.4.3.1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIVER.BEG
* Flow  Initial conditions       program version: 2.xx
*
   1[a]0.0000[b]0.0000[c]1.0000[d]1.0000[e]
   2 0.0000 0.0000 1.0000 1.0000
   3 0.0000 0.0000 1.0000 1.0000
   4 0.0000 0.0000 1.0000 1.0000
```

The following list gives explanation about the items in the file.

a number of the section or structure (+300).
b "Initial levels", at the begin of the section/structure.
c "Initial levels", at the end of the section/structure.
d "Initial discharges", at the begin of the section/structure.
e "Initial discharges", at the end of the section/structure.

Boundary conditions - Flow file

In this file all the boundary conditions related to the flow are stored. Input has been entered by "input" "flow data" "boundary conditions" (see paragraph 3.4.2.3).

Below is shown the boundary condition flow file (river.Bnd), in the appendix C (EUTROF1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIVER.BND
* Flow  Bound. cond./struct ctrl.      program version: 2.xx
*
H  a                                   2e              5b
P  c0.0000d
Q  a                                                   1b
P  c1.0000d
```

The following list gives explanation about the items in the file.

a "Type" of boundary:
  H = "Level";
  Q = "Q add.";
  QH = "Q add.Hrel";
  R = "Rain";
  W = "Wind velocity";
  T = "Wind direction".
b "Node(s)" number.
c "Type of function" = "Constant".
d Entered value for "Constant".
e Condition number.

Below an example is given, dealing with a "Level" boundary type with "Time series" specification.

```
H  a        240b  1231c     1d      2e                    1f
   1.0010g1.0020h1.0030i1.0040j1.0050k1.0060l
   1.0070m
```

The following list gives explanation about the items in the file.

a Indication "Level boundary type".
b "Time step" in minutes.
c "Start data", in yymmdd.
d "Start time", in hhmm
e condition number
f "Node(s)" number.
g 1st value of "Time series"
h 2nd value of "Time series"
: : : :

l 6th value of "Time series"
m 7th value of "Time series"

Below an example is given, dealing with a "Q add. Hrel" boundary type.

```
QH  a                                                    2b
HI  c  0.99d  0.98e  0.97f  0.96g  0.95h
QI  i1.0100j1.0200k1.0300l1.0400m1.0500n
```

The following list gives explanation about the items in the file.

a Indication "Q add. Hrel" boundary type.
b "Node(s)" number.
c Indication "Levels"
d 1st value for levels.
e 2nd value for levels.
: : : : :
g 4th value for levels.
h 5th value for levels.
i Indication "Discharges"
j 1st value for discharges.
k 2nd value for discharges.
: : : : :
m 4th value for discharges.
n 5th value for discharges.

In the example in Appendix C are no structures defined. In the part above are shown some cases, now will be given some cases of "STRUCTURE CONTROL" (see paragraph 3.4.2.4).

The first case of "STRUCTURE CONTROL" is "CONTINUOUS":

```
WIDTa                                                    302b
P   c4.9900d
SILL                                                     302
P   1.9900
GATE                                                     302
P   1.5000
MU                                                       302
P   0.9700
```

The following list gives explanation about the items in the file.

a Indication "Operational parameter" in "Continuous":
  WIDT = "Width";
  SILL = "Sill level";
  GATE = "Gate level";
  MU = "Mu (all)".
b "Structure" number (+300).
c Indication "Constant" time series.
d Value "Operational parameter".

The following example deals with "Trigger series" in the structure control.

```
TMV1[a]                                                    303[b]
TIME[c] 101010[d]     10[e]
H2H1[c]     32[f]      0[g]
H1H2[c]   2400[h]      0[g]
        34.000[i]
HPP [c]      5[j]      0[k]     33[f]
HPM          6         0        34
```

The following list gives explanation about the items in the file.

a Indication "Operational parameter" in "Trigger series":
  TSIL = "Width";
  TWID = "Sill level";
  TGAT = "Gate level";
  TMU = "Mu (all)";
  TMV1 = "Mu surf.pos.";
  TMV2 = "MU surf. neg.";
  TMO1 = "MU subm. pos.";
  TMO2 = "MU subm. neg.".

b "Structure" number (+300).

c "Type of trigger condition":
  TIME = "Time";
  H2H1 = "H2 < H1+δH";
  H1H2 = "H1 > H2+δH";
  HPP = "Hnode > Htrig";
  HPM = "Hnode < Htrig".

d Date of "Time", in yymmdd.

e Time of "Time", in hhmm.

f "Type of function" = "Constant"; new value.

g "δH"

h "Type of function" = "Time series"; "Time step" in minutes.

i 1st value of "Time series".

j "reference node"

k "trigger level"

Flow data - Result file

In this file the results from the flow calculation are printed. The concerned file is shown below. The name of the file is river.res.

```
VERSION FILE:  2.xx
ZOUT
C:\DUFLOW\RIEUT1\RIEUT1.CTR
C:\DUFLOW\RIEUT1\RIVER.NET
C:\DUFLOW\RIEUT1\RIVER.NOD
C:\DUFLOW\RIEUT1\RIVER.BND
C:\DUFLOW\RIEUT1\RIVER.BEG
C:\DUFLOW\RIEUT1\RIVER.RES
C:\DUFLOW\RIEUT1\RIEUT1.BNK
C:\DUFLOW\RIEUT1\RIEUT1.BEK
C:\DUFLOW\RIEUT1\EUTROF1.MOB
C:\DUFLOW\RIEUT1\RIEUT1.EXT
C:\DUFLOW\RIEUT1\RIEUT1.PRM
C:\DUFLOW\RIEUT1\RIEUT1.REK

ENDNAM
       0     288       0       0       4       5       3     600       0       0
       0       4       0       0       0       0       0       0       0       0
  910701       0
ENDIR0
    1      SECT   1          1       1       2    1000  -2.0000E+00  -2.0000E+00
    2      SECT   2          2       2       3    1000  -2.0000E+00  -2.0000E+00
    3      SECT   3          3       3       4    1000  -2.0000E+00  -2.0000E+00
    4      SECT   4          4       4       5    1000  -2.0000E+00  -2.0000E+00
ENDADM
       0    1   0.0000E+00   0.0000E+00   1.0000E+00   1.0000E+00   0.0000E+00
       0    2   0.0000E+00   0.0000E+00   1.0000E+00   1.0000E+00   0.0000E+00
       0    3   0.0000E+00   0.0000E+00   1.0000E+00   1.0000E+00   0.0000E+00
       0    4   0.0000E+00   0.0000E+00   1.0000E+00   1.0000E+00   0.0000E+00
       3    1   4.2339E-03   3.4274E-03   1.0000E+00   9.8963E-01   4.9646E-02
       3    2   3.4274E-03   2.6563E-03   9.8963E-01   9.9063E-01   4.9431E-02
       3    3   2.6563E-03   1.4502E-03   9.9063E-01   9.8502E-01   4.9340E-02
       3    4   1.4502E-03   0.0000E+00   9.8502E-01   9.8157E-01   4.9147E-02
       .    .       .            .            .            .            .
       .    .       .            .            .            .            .
     288    1   2.7949E-03   2.0971E-03   1.0000E+00   1.0000E+00   4.9939E-02
     288    2   2.0971E-03   1.3988E-03   1.0000E+00   1.0000E+00   4.9956E-02
     288    3   1.3988E-03   6.9971E-04   1.0000E+00   1.0000E+00   4.9974E-02
     288    4   6.9971E-04   0.0000E+00   1.0000E+00   1.0000E+00   4.9991E-02
```

Initial conditions quality file

The concerned file for the project (appendix C) is shown below. In this file all related to the quality model part are stored. The selections of the quality variable is done in "QUALITY VARIABLES FOR OUTPUT" (see paragraph 3.4.1.3). The values for these variables are entered in "INPUT" "QUALITY DATA" "INITIAL CONDITIONS" (see paragraph 3.4.2.2, 3.4.3.1).

Below is shown the initial condition quality file (RIEUT1.BEK), of the appendix C (EUTROF1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIEUT1.BEK
* Quality  Initial conditions      program version: 2.xx
*
   1 a      b  2.0000 c 2.0000 d
   2 a         2.0000 2.0000
   3 a         2.0000 2.0000
   4 a         2.0000 2.0000
   1 bod       5.0000 5.0000
   :  :           :      :
   4 bod       5.0000 5.0000
   1 nh4       0.3000 0.3000
   :  :           :      :
   4 nh4       0.3000 0.3000
   1 no3       3.0000 3.0000
   :  :           :      :
   4 no3       3.0000 3.0000
   1 norg      0.8000 0.8000
   :  :           :      :
   4 norg      0.8000 0.8000
   1 o2        10.000 10.000
   :  :           :      :
   4 o2        10.000 10.000
   1 panorg    0.0400 0.0400
   :  :           :      :
   4 panorg    0.0400 0.0400
   1 porg      0.1100 0.1100
   :  :           :      :
   4 porg      0.1100 0.1100
   1 ss        5.0000 5.0000
   :  :           :      :
   4 ss        5.0000 5.0000
```

The following list gives explanation about the items in the file.

a "Section/Structure number".
b "Variable", name of the variable.
c "Initial condition", at the begin of the section.
d "Initial condition", at the end of the section.
Etc.

Boundary conditions - Quality file

In this file all the boundary conditions related to the quality variables are stored. Input has been entered by "INPUT" "QUALITY DATA" "BOUNDARY CONDITIONS" (see paragraph 3.4.3.2).

Below is shown the boundary condition quality file (RIEUT1.BNK), of the appendix C (EUTROF1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIEUT1.BNK
* Quality  Boundary conditions      program version: 2.xx
*
C   a                                          a       b  1c
P   d1.9000e
C                                          nh4         1
P    0.3000
C                                          no3         1
P    3.0000
C                                          norg        1
P    0.8000
C                                          o2          1
P    10.000
C                                          panorg      1
P    0.0400
C                                          porg        1
P    0.1100
C                                          ss          1
P    5.0000
C                                          bod         1
P    5.0000
```

The following list gives explanation about the items in the file.

a "Type";
  C = "Concentration";
  L =.."Load".
b "Variable", selected from the picklist.
c "Node(s)", number.
d "Type of function" = "Constant".
e Value for the "Variable".

External variables file

In this file all xt external variables defined in the process description file are stored. Input has been entered by "EXTERNAL VARIABLES" (see paragraph 3.4.3.3).

Below is shown the external variables file (RIEUT1.EXT), of the appendix C (EUTROF1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIEUT1.EXT
* Quality  External variables       program version: 2.xx
*
XT  a                                                  d       b   Ac
P   d100.00e
XT  a          60f910701g     0h                       i0      b   Ac
    0.0000i0.0000 0.0000 0.0000 0.0000 1.4562

       :      :      :      :      :      :
    67.974 38.922 11.989 0.0000 0.0000 0.0000j
XT                                                     nflux       A
P   0.0500
XT                                                     pflux       A
P   0.0050
XT                                                     resf        A
P   10.000
XT              60 910701       0                      t           A
    16.333 16.243 16.163 16.087 16.003 15.930
       :      :      :      :      :      :
    21.433 21.393 21.237 20.967 20.603 20.233
XT                                                     sod         A
P   1.0000
```

a "Type";
XT = Space and time depended external variable.
b "External variable", name.
c "Node(s)", number.
d "Type of function" = "Constant".
e Value of "External variable".
f "Time step", in minutes.
g "Start data", in yymmdd.
h "Start time", in hhmm.
i First value of "Time series".
j Last value of "Time series".

Parameter file

In this file all parameters defined in the process description file are stored. The actual values have been entered by "PARAMETERS" (see paragraph 3.4.3.4).

Below is shown the parameter file (RIEUT1.PRM), of the appendix C (EUTROF1).

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\RIEUT1.PRM
* Quality  Parameters      program version: 2.xx
*
achlc [a]    30.000[b]
anc           0.100
aoc           2.670
apc           0.010
e0            2.000
ealg          0.017
fdbod         1.000
fdnorg        0.400
fdporg        0.400
fnorg         0.600
fporg         0.600
is           80.000
kbod          0.180
kbodo         0.400
kden          0.090
kdie          0.200
kdno          0.100
kmin          0.200
kmn           0.025
kn            0.025
knit          0.090
kno           2.000
kp            0.005
kpip          0.010
kres          0.125
krmin         0.200
tbzv          1.050
tden          1.045
tga           1.060
tmin          1.080
tnit          1.080
tra           1.045
trea          1.040
umax          2.100
vso           0.500
vss           1.000
```

a "Name", parameter name in the process description file.
b Entered "VALUE".

Quality data - Result file

In this file (RIEUT1.REK) all the quality results are stored.

```
VERSION FILE:  2.xx
ZOUT
C:\DUFLOW\RIEUT1\RIEUT1.CTR
C:\DUFLOW\RIEUT1\RIVER.NET
C:\DUFLOW\RIEUT1\RIVER.NOD
C:\DUFLOW\RIEUT1\RIVER.BND
C:\DUFLOW\RIEUT1\RIVER.BEG
C:\DUFLOW\RIEUT1\RIVER.RES
C:\DUFLOW\RIEUT1\RIEUT1.BNK
C:\DUFLOW\RIEUT1\RIEUT1.BEK
C:\DUFLOW\RIEUT1\EUTROF1.MOB
C:\DUFLOW\RIEUT1\RIEUT1.EXT
C:\DUFLOW\RIEUT1\RIEUT1.PRM
C:\DUFLOW\RIEUT1\RIEUT1.REK

ENDNAM
       0       96        0        0        4        5        1     1800        0        0
      12        4        0        0        0        0        0        0        0        0
  910701        0
ENDIR0
    1 a          mg-c/l      1
    2 bod        mg-o2/l     1
    3 nh4        mg-n/l      1
    4 no3        mg-n/l      1
    5 norg       mg-n/l      1
    6 o2         mg/l        1
    7 panorg     mg-p/l      1
    8 porg       mg-p/l      1
    9 chla                   2
   10 fl                     2
   11 fn                     2
   12 ft                     2
ENDVAR
    1      SECT   1          1       1       2   1000  -2.0000E+00  -2.0000E+00
    2      SECT   2          2       2       3   1000  -2.0000E+00  -2.0000E+00
    3      SECT   3          3       3       4   1000  -2.0000E+00  -2.0000E+00
    4      SECT   4          4       4       5   1000  -2.0000E+00  -2.0000E+00
ENDADM
       0    1   2.0000E+00   2.0000E+00   2.0000E+00   2.0000E+00   0.0000E+00
       0    1   5.0000E+00   5.0000E+00   5.0000E+00   5.0000E+00   0.0000E+00
       0    1   3.0000E-01   3.0000E-01   3.0000E-01   3.0000E-01   0.0000E+00
       0    1   3.0000E+00   3.0000E+00   3.0000E+00   3.0000E+00   0.0000E+00
       0    1   8.0000E-01   8.0000E-01   8.0000E-01   8.0000E-01   0.0000E+00
       0    1   1.0000E+01   1.0000E+01   1.0000E+01   1.0000E+01   0.0000E+00
       0    1   4.0000E-02   4.0000E-02   4.0000E-02   4.0000E-02   0.0000E+00
       0    1   1.1000E-01   1.1000E-01   1.1000E-01   1.1000E-01   0.0000E+00
       0    1   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00
       0    1   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00
       0    1   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00
       0    1   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00
       0    2   2.0000E+00   2.0000E+00   2.0000E+00   2.0000E+00   0.0000E+00
       0    2   5.0000E+00   5.0000E+00   5.0000E+00   5.0000E+00   0.0000E+00
       0    2   3.0000E-01   3.0000E-01   3.0000E-01   3.0000E-01   0.0000E+00
       0    2   3.0000E+00   3.0000E+00   3.0000E+00   3.0000E+00   0.0000E+00
       0    2   8.0000E-01   8.0000E-01   8.0000E-01   8.0000E-01   0.0000E+00
       0    2   1.0000E+01   1.0000E+01   1.0000E+01   1.0000E+01   0.0000E+00
```

0	2	4.0000E-02	4.0000E-02	4.0000E-02	4.0000E-02	0.0000E+00
0	2	1.1000E-01	1.1000E-01	1.1000E-01	1.1000E-01	0.0000E+00
0	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	3	2.0000E+00	2.0000E+00	2.0000E+00	2.0000E+00	0.0000E+00
0	3	5.0000E+00	5.0000E+00	5.0000E+00	5.0000E+00	0.0000E+00
0	3	3.0000E-01	3.0000E-01	3.0000E-01	3.0000E-01	0.0000E+00
0	3	3.0000E+00	3.0000E+00	3.0000E+00	3.0000E+00	0.0000E+00
0	3	8.0000E-01	8.0000E-01	8.0000E-01	8.0000E-01	0.0000E+00
0	3	1.0000E+01	1.0000E+01	1.0000E+01	1.0000E+01	0.0000E+00
0	3	4.0000E-02	4.0000E-02	4.0000E-02	4.0000E-02	0.0000E+00
0	3	1.1000E-01	1.1000E-01	1.1000E-01	1.1000E-01	0.0000E+00
0	3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	4	2.0000E+00	2.0000E+00	2.0000E+00	2.0000E+00	0.0000E+00
0	4	5.0000E+00	5.0000E+00	5.0000E+00	5.0000E+00	0.0000E+00
0	4	3.0000E-01	3.0000E-01	3.0000E-01	3.0000E-01	0.0000E+00
0	4	3.0000E+00	3.0000E+00	3.0000E+00	3.0000E+00	0.0000E+00
0	4	8.0000E-01	8.0000E-01	8.0000E-01	8.0000E-01	0.0000E+00
0	4	1.0000E+01	1.0000E+01	1.0000E+01	1.0000E+01	0.0000E+00
0	4	4.0000E-02	4.0000E-02	4.0000E-02	4.0000E-02	0.0000E+00
0	4	1.1000E-01	1.1000E-01	1.1000E-01	1.1000E-01	0.0000E+00
0	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
0	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	1	1.9762E+00	1.9862E+00	1.9762E+00	1.9409E+00	0.0000E+00
1	1	4.9974E+00	4.9977E+00	4.9974E+00	4.8835E+00	0.0000E+00
1	1	3.0261E-01	3.0299E-01	3.0261E-01	2.9606E-01	0.0000E+00
.	.	.	.	.	.	.
.	.	.	.	.	.	.
96	4	6.0750E+01	6.1093E+01	0.0000E+00	0.0000E+00	0.0000E+00
96	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
96	4	9.0726E-01	9.0795E-01	0.0000E+00	0.0000E+00	0.0000E+00
96	4	1.1049E+00	1.1049E+00	0.0000E+00	0.0000E+00	0.0000E+00

Measured data file

In this file (MEASUR.MSR) the measured data can be stored. These data has be edited in "Input" "measured Data". In the file shown below is an example given of a measured value. Chosen is the variable ss, with a constant value of 10.000.

The format of this file is equal to that of the boundary condition file.

```
* DUFLOW data file :C:\DUFLOW\RIEUT1\MEASUR.MSR
* Measured data      program version: 2.xx
*
C                                                ss        5
P    10.000
```

Appendix F Errors in duflow

In this appendix a list of possible errors are given which can occur in the program DUFLOW. The explanation of the error is given and also a possible solution.
If you have any question about the working of program DUFLOW, you can always consult the EDS Helpdesk (070-3014600).

Errors	Explanation
Out of memory, or Out of string space Internal memory full	The program is trying to allocate too much memory, see «Setup» «Maximum data size». The input-program tries to store more data than the allocated amount of memory allows, see «Setup» «Maximum data size»
Internal memory error	This error can only occur due to a bug in the input program. If so, please report it to the EDS Helpdesk.
Invalid argument(s)	Contact EDS.
File <fname> not found	Model-file not found.
Unexpected end of file	Probably a missing }.
Line too long	Lines must be 100 characters long or shorter.
Character not allowed	Was expecting a number.
Numbers not allowed	Was expecting an identifier.
Invalid identifier	An identifier consists of 6 characters or numbers and must start with a character.
Invalid number	
Missing]	Missing [.
Missing range	A identifier is declared more than once.
Multiple declaration	
Missing =	The identifier in the k1 or k0 function is not a state variable.
Not a state variable	
Identifier not declared	Was expecting an identifier, found an operator.
Missing identifier/number	Was expecting an operator, found an identifier.
Missing operator	Disk full.
Cannot open object file	else-statement without a preceding if-statement
Dangling else-statement	identifier is not a flow variable
Invalid flow variable	if-statements not allowed in functions
Misplaced if-statement	else-statements not allowed in functions
Misplaced else-statement	iterate-statements not allowed in functions
Misplaced iterate-statement	Invalid iterate syntax see chapter 3.
Was expecting iteration-variable	Invalid iterate syntax see chapter 3.
Was expecting iteration tolerance	Check network.
Ambigious section number	Check network.
Begin node equals end node	Contact EDS.
Both streaming widths or areas	Check boundary conditions, external variables or locations for output.
Error in n/s string	Check if: - The computation period does fit into the time-series; - The date/time is ascending (non equidistant time-series).
Error in timeseries	
Error while opening <File-name>	The disk is full.
Error while opening dumpfile	The disk is full.

Error while opening ...temp. boundary file (flow)	The disk is full.
Error while opening ...temp. boundary file (quality)	The disk is full.
Error while opening ...temp. external vars. file	The disk is full.
Error while opening temp. units file	The disk is full.
Error while writing temp. unitsfile	The disk is full.
Error writing scratchfile	The disk is full.
Incomplete section data	Check network.
Instable computation timestep NNN	Waterlevel exceeded 1000 m.
Internal error memory-manager	Contact EDS.
Invalid code	Cont act EDS.
Invalid modeldefinitionfile	Contact EDS.
Invalid node	Check boundary-conditions, external variables or locations for output.
Invalid section	Check boundary-conditions, external variables or locations for output.
Invalid section or node	Check boundary-conditions, external variables or locations for output.
Missing control card	Contact EDS.
Missing time card	Contact EDS.
No streaming widths or areas	No streaming width or areas given.
Node does not exist	Check boundary-conditions, external variables or locations for output. Not enough memory available. The problem is too big for DUFLOW (use DUFDIM to examine the problem).
Pivot-element NN is zero	Contact EDS.
Read error	Contact EDS.
Section does not exists	Check boundary-conditions, external variables or locations for output.
Section is not a structure	Check triggers.
Syntax error	Contact EDS.
Triggercondition has 0 triggers	Check triggers.
Unexpected end of dumpfile	Invalid dumpfile used.
Wrong code	Contact EDS.

IHE DELFT

P.O. Box 3015
2601 DA Delft
The Netherlands

Tel. : +31(0)15 2151715
Fax : +31(0)15 2122921
E-mail: ihe@ihe.nl