Report on the ICES subsystem FLOWS

by N. Booij

Report no. 78-3

Parts of this report have been published in slightly different form previously in:


Abstract

FLOWS is a recently developed ICES subsystem for applications in the field of hydraulic engineering. It is able to compute steady as well as unsteady flows in hydraulic networks of given dimensions. This means that it does not optimize any parameters describing the network. The network may be composed of open and closed channels, reservoirs and flow control devices such as pumps, weirs etc. This covers water supply networks, estuaries, river and canal networks etc.

Besides this, FLOWS computes the transport of pollutant in a hydraulic network. The method is restricted to such problems where the concentration is nearly uniform over the cross-section and where density differences have a negligible effect on the flow. Possible applications are cooling water circuits, salt intrusion etc.

Some experience with respect to the flow computations is reported. Appendices give details about the numerical approximation and its accuracy, and about the use of the subsystem.
## Contents

1. Introduction 2
2. Sample Problems 3
3. Overview of subsystem's facilities 6
4. Choice of the numerical scheme 8
5. Some experience and conclusion 10

Acknowledgement 13

References 14

List of symbols 16

## Appendices

A. The numerical scheme employed in FLOWS 18
B. Accuracy 33
C. User's Guide. 45
1. **Introduction**

The earliest applications of electronic computers in the field of flow computations have been in tidal computations, and in computations of water distribution networks. Tidal flows in the beginning were computed mostly by means of explicit numerical schemes (see e.g. **Dronkers**, ref. 1). Such schemes have the disadvantage that a rather strict limit is imposed on the time step, in order to maintain stability. On the other hand they lead to simple computer programming. For tidal computations even now the explicit method is fairly efficient. Later on implicit schemes have been promoted, for instance by **Amein** (ref. 2) and **Abbott** (ref. 3). This trend is connected with the increasing use of computers for problems of water management. Such processes are characteristically slower than tidal processes, and consequently extend over larger periods. Then explicit schemes are less efficient due to their limited time step. Moreover in water management more emphasis nowadays is put on water quality problems, which again puts implicit methods in a better position.

In the Department of Civil Engineering of the Delft University of Technology a computer program for tidal computations is available, but in view of the development mentioned, it was decided to build up an integrated system for both water flows and transports of pollutant, not only in single channels, but also in networks. The latter requirement is important especially in a country like the Netherlands, where complicated networks of canals and rivers exist. Fig. 1 shows an example, which is already a considerable simplification of reality.

It was decided to write the package as a subsystem of ICES, the Integrated Civil Engineering System, developed originally at M.I.T. (Ref. 12). The conventional programming languages such as Fortran or Algol offer too little flexibility for complex application
packages such as FLOWS. Facilities of ICES as the problem oriented input language, dynamic arrays and the dynamic loading of subroutines are used widely in FLOWS.

2. Sample problems

The class of problems which the subsystem is intended to solve, can be indicated by an enumeration of sample problems. Such problems originate from water management, tidal engineering, flood control, water distribution etc.

Since flows in water distribution networks are usually assumed to be steady, the computation of steady flows has to be included in the package. A summary of the input for the FLOWS subsystem would be:

```
FLows
data describing the network
data defining the boundary conditions
command: compute steady flows
FINISH
```

Estuaries, the study objects of tidal engineering, are of many different shapes. Some of them are single channels, some consist of networks of channels, and some are essentially two-dimensional, and can therefore not be handled by the FLOWS package. The flows are always unsteady, and often periodic. In order to start a computation of unsteady flows one needs initial values of both discharges and water levels. These are not known beforehand, but that is not harmful, since in the periodic case one can start with an assumed initial condition and then compute over such a long time span, that the whole flow pattern has become periodic. At that moment the influence of the initial condition has disappeared. The input summary looks like:
FLOWS
data describing the network
data defining the boundary conditions
assumed initial values
command: COMPUTE UNSTEADY FLOWS OVER .. STEPS
FINISH

In flood control, on the contrary, the initial condition is more or
less known, it can often be assumed to be a steady flow in the river
under consideration. So the package FLOWS should be able to compute
a steady flow in a river or network of rivers, and then use this as
the initial condition for an unsteady flow computation.
The input summary looks like:
FLOWS
data describing the network
(steady) boundary conditions
command: COMPUTE STEADY FLOWS
(unsteady) boundary conditions
command: COMPUTE UNSTEADY FLOWS ..
FINISH

As far as flow computations are concerned, the cases mentioned also
may arise in problems of water management. Here, however, one often
is confronted with water quality problems as well. The following
types of problems can occur:
- steady flow with steady concentration of pollutant: an example is
  an electric power plant which carries waste heat at a steady rate
  into a steadily flowing river.
- steady flow with unsteady concentrations; this may happen when
  the same river is polluted at an unsteady rate.
- unsteady flows and concentrations; such as one finds if the elec-
  tric power plant is situated along an estuary.
It is impossible, except in a trivial case, to have steady concen-
trations and unsteady flows. In the case of the power plant on the
estuary, the water temperature is unsteady also if the waste heat is deposited at a constant rate. The trivial case referred to, is the one with equal concentration all over the network which are also constant with time.

For the three types of problems the input summaries are:

**FLows**

- Data describing the network
- Boundary conditions (for flows and concentrations)
- Command: COMPUTE STEADY FLOWS AND CONCENTRATIONS
- Finish

**FLows**

- Data describing the network
- Boundary conditions (for water flow)
- Command: COMPUTE STEADY FLOWS
- Boundary conditions (for concentrations)
- Initial conditions (for concentrations)
- Command: COMPUTE UNSTEADY CONCENTRATIONS
- Finish

**FLows**

- Description of the network
- Boundary conditions (for flows and conc.)
- Initial conditions (id.)
- Command: COMPUTE UNSTEADY FLOWS AND CONCENTRATIONS

Due to the present state of knowledge it was decided that FLOWS would not be able to compute variations in the parameters describing the networks, such as widths or bottom levels. This means that is does not optimize any of these parameters, nor does it compute their variations due to sediment transport. In other words it makes computations with fixed bed. The question is considered whether the latter restriction could be abolished.
3. **Overview of the subsystem's possibilities**

Branches of type canal, pipe or river usually form the bulk of a network model. FLOWS can accommodate these, each of them with several variations. Canals can be horizontal or inclined, pipes can be either permanently full or with cavitating flow, they may or may not have an elastic storage. So far, only circular pipes can be applied. River branches can have different types of cross-sections, they may or may not be shallow, they can have a storage width equal to the stream width, or not equal.

Moreover the user can choose between three different types of friction laws.

As a rule a model needs some boundary value equipment. FLOWS knows three types of boundary elements: constant, varying according to a given set of values, periodic based on Fourier coefficients. Water level, discharge, concentration and transport can serve as boundary value.

As FLOWS came into use, a need developed for elements such as reservoirs (pure storage elements) and for flow control elements such as pumps, weirs, and stage-discharge relations.

This led to the introduction of a hybrid element, consisting of a reservoir and/or a weir, pump etc. This element has been tested and is now in use.

Types of elements that may be introduced in the future, are: groundwater flow, in either confined or unconfined aquifers; polder or other catchment elements, relating rainfall data to a discharge into a river or canal network.

Standard output by FLOWS is the printing of the state of the network at regular intervals.

Upon user's request other outputs can be made. Printed plots of selected variables can be produced, and values of selected variables
can be written on disk, tape or cards for use as input to other programs. This facility can also be helpful in finding modelling errors. However, more facilities have to be added, such as plotting of the network, in order to check its geometry.

Some other more radical extensions of the subsystem may prove to be necessary. One would be the coupling with two-dimensional flow such as one finds in lakes, bays and coastal seas. Another is the description of water quality by more than one parameter. Such extensions would bring about serious modifications of the existing subsystem. It is thought that the priority of such improvements should be settled on the basis of practical experience.

Appendic C, the user's guide, gives more detailed information on the presently available possibilities.
4. Choice of the Numerical Scheme

As a consequence of the considerations given before, the numerical scheme that had to be chosen, should satisfy at least the following requirements:
- no strict limit on the time step,
- suitable for networks,
- suitable for steady as well as unsteady computations,
- data describing the water flow have to be used as input for the water quality computations.

The first requirement excludes explicit schemes, so that implicit schemes remain to be considered. There are two widely used types, one where values of discharge and of water level are taken at alternating points along the channel, the other with these variables at the same points (see e.g. Dronkers, ref. 4).

The first of these two leads to somewhat simpler programming, and is a little less expensive, in terms of computer cost. The latter has the advantage that spatial steps can be varied without loss of accuracy. This is important for network computations, because in networks the schematization can not always be chosen such that the steps are equal or even nearly equal. The method is valid for unsteady as well as steady flow. In the case of steady flow the method resembles the one described by Marlow, Hardison, Jacobson and Biggs (ref. 5), who construct a set of linearised nodal equations, which is solved by means of an iterative method of solution, whereas this paper uses a direct method. A solution method for sparse matrices is used in order to obtain the solution in an economical way.

It appears that a similar numerical scheme is suitable for the water quality computation itself. The scheme fits to the requirements mentioned above, with the exception of the one pertaining to networks. Near branching points extremely small spatial steps would be needed,
due to the small value of the diffusion coefficient in the equation for the water quality parameter. This drawback is removed if the numerical scheme is changed in such a way that discontinuities are allowed at the nodal points. Details of the computational method are found in appendix A.

The most important restriction of the numerical method employed here, is that supercritical flow is not allowed. Such a flow occurs in mountain rivers, and sometimes in sewer networks.
5. Some experience and conclusions

In the region Friesland in the Netherlands an extensive canal network exists which carries excess water from the polders to the sea or the Yssel Lake. Flow computations have been made in order to improve the water management in the area.

A lot of the smaller canals have been left out in the computation but still a network remains of over 100 branches. It is depicted in fig. 1. Since the wind is an important factor influencing the water levels, two periods, one with relatively mild wind, and one with strong wind have been simulated. In the centre of the area the correspondence with measured data was quite good, at the outskirts of the area it was less, with deviations up to 10 cm.

It was felt, however, that with adequate trimming of the model, the deviation can be reduced considerably. Such work was outside the scope of this study.

For problems of the kind mentioned above a large amount of data has to be punched. Mistakes are likely to be made, so a good error detection system is indispensable. Improvements in this field have still to be made. Also the amount of data could be reduced somewhat if more was known about the relationship between rainfall and the flow from polders onto the canal system. Research is being done to establish such relationships.

Another application has been the flood routing in a river. One study has been done on a theoretical river, with the aim of checking the accuracy of the computational scheme. The accuracy had also been investigated by means of the analysis of the propagation of sinusoidal waves in a linearized model. Such analysis had been done earlier for waves with dominating inertia. In flood waves, however, the friction term is dominant over the inertia, so that a different analysis had to be done. The results of both analyses are found in appendix B. In one respect the FLOWS subsystem failed so far. If a flood wave
in a river is studied, usually a steady state is used as initial state. This ought to be done using the command 'COMPUTE STEADY FLOWS ...', but in the case of a river this produced divergent numbers. Therefore, the steady state has so far been computed by means of the command 'COMPUTE UNSTEADY FLOWS ...' together with steady boundary conditions and choosing the number of time steps sufficiently large to reach a steady state.

FLOWS has also been applied to a more realistic river problem, connected with the Rio Bogota in Colombia. In this case the numerical model has been compared with a scale model. Both models agreed satisfactorily.

As yet, the concentration computation has not been tested on practical problems. There are plans to start with this in the near future.
Fig. 1. Network of canals and lakes in the region Friesland.
Acknowledgement

The author wishes to thank Mr. K. Popp for his contribution in the research on the solution of linear systems, Mr. M.W. Rinkel for his help in the examination of several finite difference methods for the water quality computation, and some students of the Delft University who were the first to use FLOWS for practical problems and who thereby ran the risk of many inconveniences. Among them Mr. A. Tuinhof is to be mentioned explicitly.
References


List of Symbols

A  flow cross-section
b  width of storage = dB/dh
B  total wetted cross-section
C  concentration
C_k concentration in node k
C_1, C_2 concentrations at both ends of a branch
d depth
D  diffusion coefficient
e  base of natural logarithms
g  acceleration of gravity
h, H water level
i  \sqrt{-1}
I  bottom slope
I(m) node at one end of branch m
J  slope due to friction J = J(Q,h)
J(m) node at second end of branch m
K  structure matrix J(m) node at second end of branch m
L  spatial step size (= length of a branch)
L  wave length
M  matrix element
n_c ratio of wave celerity in discrete and continuous model
n_d ratio of wave damping in discrete and continuous model
N  submatrix
P  production term in the transport equation
Q  discharge
r  a complex number
R  hydraulic radius
s  a complex number
S  transport of pollutant
t  time
T  wave period
v  water velocity
w  width of flow = dA/dh
x  coordinate along channel axis
y  perturbation of depth
z  bottom level
θ  weighting factor, controls numerical damping
π  circular constant (3.1416)
τ  time step

subscripts 1,2 refer to both ends of a branch
subscripts -, + refer to consecutive time levels or to consecutive iterations.
Appendix A. Numerical Schemes employed in FLOWS

A.1. The representation of networks

The hydraulic network is schematized to a graph; a graph consists of branches and nodes, called edges and vertices in graph theory. In fig. A.1. a network of open channels is sketched together with its schematization.

Each branch connects two nodes, and the most simple way to describe the graph is by means of the associated list: to all nodes and branches numbers are assigned, the numbering of the nodes and the numbering of the branches being independent of each other. The associated list contains the numbers of the nodes that are connected by each branch. For branch m these nodes will be denoted I(m) and J(m).

Fig. A.1.
The computational procedure for flows and pollutant transport is identical. Linearised relations between discharges and water levels, or between transport and concentrations are established. The resulting set of equations is solved, and if non-linearity is important, the procedure is repeated one or more times. For each branch two equations are derived, one is based on the continuity equation, the other on the equation of motion. For each node one equation, based on the continuity equation, is found. After elimination of all discharge values (or transport values) a set of equations remains in which only water levels (or concentrations) appear, viz.

(A.1) \[ \sum_{j} M_{ij} H_j + M_{io} = 0 \]
A.2. The numerical model for water flow

For flow in one-dimensional channels a set of two equations is used. One of these is the momentum equation.

\[(A.2) \quad \frac{\partial Q}{\partial t} + \frac{\partial (Qv)}{\partial x} + gA \frac{\partial h}{\partial x} + J(Q,h) = 0\]

Q indicates discharge, v velocity, A the flow cross-section, and J is the slope due to friction. Secondly one has the continuity equation:

\[(a.3) \quad \frac{\partial B}{\partial t} + \frac{\partial Q}{\partial x} = 0\]

B is the total wetted cross-section, as a rule some given function of the water level \(h\), \(b = dB/dh\) is the width of storage (see fig. A.2).

![Diagram](image)

**Fig. A.2.** Definition sketch of channel cross-section.

Each branch of the network is considered as one elementary step in the numerical analysis. Often one branch has to be divided in shorter branches for the sake of accuracy. The question of accuracy is considered in more detail in appendix B.
The discharge and water level in each branch are characterized by their values at both ends of the branch. The discharge values for branch numbered m, are $Q_{1,m}$ and $Q_{2,m}$, the water levels $h_{1,m}$ and $h_{2,m}$. These water levels are identical to the water levels in the adjacent nodes, whose numbers are I(m) and J(m) (see fig. A.3). If the level in node i is called $H_i$, we have the relations:

(A.4) \[ h_{1,m} = H_I(m) \]
\[ h_{2,m} = H_J(m) \]

Fig. A.3. One branch and its adjacent nodes.

In the case of unsteady flow one further distinction has to be made, viz. between different time levels. Therefore a superscript $+$ is introduced to indicate time $t + \tau$ ($\tau$ is time step) and a superscript $-$ indicating time $t$ (see fig. A.4).

Fig. A.4.
As point of reference a point near the center of the rectangle in fig. A.4. is chosen: \( (\frac{1}{2}(x_1 + x_2), t + \theta t) \). The weighting factor \( \theta \) can generate numerical damping, if it is larger than 0.5 (see for instance Ref. 10 or 13). The partial derivatives of some variable \( \phi \) are approximated by:

\[
\frac{\partial \phi}{\partial t} = \frac{\phi^+ + \phi^- - \phi^+ - \phi^-}{2\tau}
\]

\[
\frac{\partial \phi}{\partial x} = \theta \left( \frac{\phi^+ - \phi^-}{\lambda} \right) + (1 - \theta) \left( \frac{\phi^+ - \phi^-}{\lambda} \right) + \ldots
\]

and a function of \( \phi \) by:

\[
F(\phi) = F \left( \phi \left( \frac{x_1 + x_2}{2}, t + \theta t \right) \right) = \frac{1}{2} \left\{ F(\phi^+) + F(\phi^-) + \theta \left( \frac{\partial F}{\partial \phi} \right)_1 (\phi^+ - \phi^-) + \theta \left( \frac{\partial F}{\partial \phi} \right)_2 (\phi^+ - \phi^-) \right\}
\]

\( \lambda \) is the length of the branch. After substituting these approximations into eqs. (A.2) and (A.3) one obtains resp.

\[
\frac{Q^+_{1} - Q^-_{1} - Q^+_{2} - Q^-_{2}}{2\tau} + \frac{v^+_{2} Q^+_{2} - v^-_{1} Q^-_{1}}{\lambda} + \frac{g(A^+_{1} + A^-_{2})}{2\tau} \left( \theta h^+_2 - \theta h^-_1 + (1 - \theta) (h^-_2 - h^-_1) \right) + \frac{1}{2} \left\{ J(Q^+_{1}, h^-_1) + \theta \frac{\partial J}{\partial Q} (Q^+_{1} - Q^-_{1}) + \theta \frac{\partial J}{\partial h} (h^+_1 - h^-_1) \right\} + J(Q^-_{2}, h^-_2) + \theta \frac{\partial J}{\partial Q} (Q^+_{2} - Q^-_{2}) + \theta \frac{\partial J}{\partial h} (h^+_2 - h^-_2) = 0
\]

and

\[
\frac{B^+_1 + b^+_1 h^+_1 + B^+_2 + b^+_2 h^+_2 - B^-_1 - B^-_2}{2\tau} + \theta \frac{Q^+_{2} - Q^-_{1}}{\lambda} + (1 - \theta) \frac{Q^-_{2} - Q^-_{1}}{\lambda} = 0
\]
Here $B_1^* = B_1^+ - b_1^+ h_1^-$. After the values of $h$ have been calculated $B_1^+$ is set equal to $B_1^* + b_1^+ h_1^+$. Obviously $B_1^*$ and $b_1^+$ are not known in advance, since they depend on $h_1^+$, but they can be estimated and they can be improved in an iteration process. In this numerical model the water quantity is exactly conserved, apart from rounding errors.

Eq. (A.8) and (A.9) form a set of two equations with four unknowns, viz. $Q_1^+, Q_2^+, h_1^+, h_2^+$. These equations are linear with respect to each of these unknowns, so that they can be transformed into a set of equations in which both $Q$'s are written explicitly.

In short this set is written

\[
\begin{align*}
Q_{1,m}^+ &= N_{m,1} H^+ I(m) + N_{m,2} H^+ J(m) + N_{m,3}, \\
Q_{2,m}^+ &= N_{m,4} H^+ I(m) + N_{m,5} H^+ J(m) + N_{m,6}.
\end{align*}
\]

Another set of equations derives from the continuity in each node. Because no storage is allocated to the nodes, these equations are simply $\sum Q = 0$. The sign of each $Q$ has to be taken into account; this is accomplished by introducing matrices $K_1$ and $K_2$ related to the structure of the network:

\[
\begin{align*}
K_1(i,m) &= 1 \quad \text{if} \quad i = I(m) \\
K_1(i,m) &= 0 \quad \text{otherwise} \\
K_2(i,m) &= 1 \quad \text{if} \quad i = J(m) \\
K_2(i,m) &= 0 \quad \text{otherwise}
\end{align*}
\]

The continuity equations for the nodes are now written more precisely:

\[
\sum_{m}(K_2(i,m) Q_{2,m} - K_1(i,m) Q_{1,m}) = 0
\]
Substituting (A.10) into (A.12) one eliminates the Q's and ends up with a set of linear equations with the H's as unknowns in eq. (A.1).

The procedure for building up the system of linear equations is described in more detail in sec. A.4.

The case of steady flow is treated in the same way. In the notation there appear again $\phi^-$ and $\phi^+$, however the meaning of these symbols is different. '-' and '+' refer to two consecutive iteration steps, instead of time levels. The continuity equation reduces to:

(A.13) \[ Q_1^+ = Q_2^+ = Q_0^+ \]

Also in the equation of motion all time derivatives vanish. There remains

(A.14) \[ \frac{v_2^- - v_1^-}{l} Q^+ + g \left( \frac{A_1^- + A_2^-}{2l} (h_2^+ - h_1^+) \right) + \]

\[ + \frac{1}{2} \left( J(Q^-, h_1^-) + \frac{\partial J}{\partial Q} (Q^+ - Q^-) + \frac{\partial J}{\partial h} (h_1^+ - h_1^-) \right) + \]

\[ + J(Q^-, h_2^-) + \frac{\partial J}{\partial Q} (Q^+ - Q^-) + \frac{\partial J}{\partial h} (h_2^+ - h_2^-) \] \[ = 0 \]

This leads to equations of the form (A.10).

So from this point on the solution method is the same as in the unsteady case.
A.3. Computation of concentrations

The flow in the hydraulic network is supposed to be well-mixed so that concentrations in one profile can be characterized by one number $C$. If the transport rate is denoted by $S$ and the production of pollutant by $P$, the continuity equation looks like

\begin{equation}
\frac{\partial S}{\partial x} + \frac{\partial B C}{\partial t} - P = 0
\end{equation}

If only convection is regarded, $S = Q C$. The effect of dispersion is imitated by adding a term $-A D \frac{\partial C}{\partial x}$, which may sometimes be negligible, sometimes not. So

\begin{equation}
S = Q C - A D \frac{\partial C}{\partial x}
\end{equation}

A consequence of this model is that in regions where strong gradients of the concentration occur, the transport can be in the direction opposite to the water flow. In practice, however, no upstream influence is usually expected. The fact that $D$ is often very small, causes a very strict limit on the magnitude of the distance step. This limitation is absent if the numerical model excludes upstream influence.

The numerical model that will be examined here, is based on the Galerkin criterion (Refs. 6 and 7)

\begin{equation}
\int_{0}^{L} \psi_i(x) \left( \frac{\partial S}{\partial x} + \frac{\partial (B C)}{\partial t} - P \right) \, dx = 0 = \\
\left. \psi_i S \right|_{0}^{L} + \int_{0}^{L} \left( \psi_i \frac{\partial (B C)}{\partial t} - \frac{\partial \psi_i}{\partial x} (Q C - A D \frac{\partial C}{\partial x}) - \psi_i P \right) \, dx = 0
\end{equation}

The trial functions are $\psi_1 = 1 - x/L$ and $\psi_2 = x/L$. Then $C = c_1 \psi_1(x) + c_2 \psi_2(x)$. Working out the integral (A.17) for $i = 1, 2$ leads to
(A.18) \[ -S_1 + \frac{2}{3} \frac{\partial(B_1c_1)}{\partial t} + \frac{\partial(B_2c_2)}{\partial t} + \frac{Q_1c_1 + Q_2c_2}{2} + \]
\[ -AD \left( \frac{c_2 - c_1}{\ell} - \frac{2}{3} \ell P_1 - \frac{5}{6} \ell P_2 \right) = 0 \]

(A.19) \[ S_2 + \frac{\ell}{6} \frac{\partial(B_1c_1)}{\partial t} + \frac{\partial(B_2c_2)}{\partial t} - \frac{Q_1c_1 + Q_2c_2}{2} + \]
\[ + AD \left( \frac{c_2 - c_1}{\ell} - \frac{2}{3} \ell P_1 - \frac{5}{6} \ell P_2 \right) = 0 \]

The subsequent discretization with respect to time necessitates a distinction between \(c_1^-\) and \(c_1^+\) etc., as in the previous section. Then

(A.20) \[ S_1^+ = -S_1 - 2\ell \frac{B_1^{+c_1^+} - B_1^{c_1^-}}{\tau} + \frac{1}{3} \ell \frac{B_2^{+c_2^+} - B_2^{-c_2^-}}{\tau} + \]
\[ + \frac{Q_1c_1^+ + Q_2c_2^+ + Q_1c_1^- + Q_2c_2^-}{2} - AD \left( \frac{c_2^+ + c_2^- - c_1^+ - c_1^-}{\ell} \right) - \frac{2}{3} \ell P_1 - \frac{1}{3} \ell P_2. \]

(A.21) \[ S_2^+ = -S_2 - \frac{1}{3} \ell \frac{B_1^{+c_1^+} - B_1^{c_1^-}}{\tau} - \frac{2}{3} \ell \frac{B_2^{+c_2^+} - B_2^{-c_2^-}}{\tau} + \]
\[ + \frac{Q_1c_1^+ + Q_2c_2^+ + Q_1c_1^- + Q_2c_2^-}{2} - AD \left( \frac{c_2^+ + c_2^- - c_1^+ - c_1^-}{\ell} \right) + \]
\[ + \frac{1}{3} \ell P_1 + \frac{2}{3} \ell P_2. \]

A distinction has now to be made between the concentration inside the branch, characterized by \(c_1\) \text{ and } \(c_2\), the concentrations at both ends, and the concentrations in the nodes \(I(m)\) and \(J(m)\), being the nodes at both ends. The latter concentrations are called \(C_I\) \text{ and } \(C_J\). In order to exclude upstream transport, it may be necessary to drop the assumption that \(c_1 = C_I\) or \(c_2 = C_J\). The assumption is maintained at the end of the branch where the flow is directed inward; if the flow is directed outward, the concentrations are not necessarily identical and the transport at that end is assumed to be the flow rate times the concentration at that end of the branch.
(A.22) \[ c_1^+ = C_1^+, \text{ if } Q_1^+ \geq 0, \]
\[ S_1^+ = Q_1^+ c_1^+, \text{ if } Q_1^+ < 0, \]
\[ c_2^+ = C_2^+, \text{ if } Q_2^+ \leq 0, \]
\[ S_2^+ = Q_2^+ c_2^+, \text{ if } Q_2^+ > 0. \]

As an example suppose \( Q_1^+ \) and \( Q_2^+ \) to be positive. Then \( S_2^+ = Q_2^+ c_2^+ \), and this together with eq. (A.21) enables us to find a relation between \( c_2^+ \) and \( c_1^+ \), or between \( S_2^+ \) and \( c_1^+ \), and since \( c_1^+ = C_1^+ \), also between \( S_2^+ \) and \( C_1^+ \). Eq. (A.20) then gives a relation between \( S_1^+ \) and \( C_1^+ \). This way one gets equations fully analogous to eq. (A.10), viz.

(A.23) \[ S_1^+ = N_{m,1}^+ C_1^+ + N_{m,2}^+ C_1^+ + N_{m,3}^+ \]
\[ S_2^+ = N_{m,4}^+ C_1^+ + N_{m,5}^+ C_1^+ + N_{m,6}^+. \]

As in the case of water flow, in each node one has \( \sum S = 0 \), which leads to a set of equations

(A.24) \[ \sum_{i,j} N_{i,j}^+ C_{j}^+ + M_{i,10}^+ = 0. \]

In the example described before (pos. \( Q_1^+ \) and \( Q_2^+ \)) the coefficients \( N_{m,2}^+ \) and \( N_{m,5}^+ \) vanish. This has important consequences for the solution strategy for the set of linear equations. Generally, if \( Q_1^+ \) is negative, \( N_{m,1}^+ = N_{m,4}^+ = 0 \), and if \( Q_2^+ \) is positive, \( N_{m,2}^+ = N_{m,5}^+ = 0 \). The assumptions (A.22) can only be used if small values of \( D \) are considered. If \( D \) is large, it is possible that upstream transport does occur, and then we always suppose \( c_1^+ = C_1^+ \) and \( c_2^+ = C_2^+ \), irrespective of the signs of \( Q_1^+ \) and \( Q_2^+ \). Then none of the coefficients \( N \) can be expected to be zero.

Since, to the writer's knowledge, this numerical scheme has not been studied elsewhere, it has to be tested thoroughly. The results of some simple tests are shown below. They pertain to a river with uniform water velocity where at \( x = 0 \) for \( t \geq 0 \) an input of pollu-
tant is simulated. At $t = 0$ the river is free of pollutant. There is no production or destruction, so $P = 0$. All parameters like $Q, A, B, D$, etc. are constant. Further $A = B = \text{const.}$ and $v = Q/A$. This problem is fully described by the dimensionless parameters $v' = v\tau/\lambda$ and $D' = D\tau/\lambda^2$. As fig. A.5 shows, there is some overshoot, if $D' = 0$. This overshoot grows for growing $v'$. Fig. A.6 shows that even a rather small value of $D'$ makes the overshoot disappear. For larger values of $D'$ the discontinuities that are to be expected due to the uncoupling of the concentrations, become more and more apparent. However, it has already been said, for such values the completely coupled model should be used. If this is used, the picture of fig. A.7 is obtained instead.

![Graph](image_url)  

Fig. A.5.
Fig. A.6.

Fig. A.7.
A.4. **Solution of the linear equations**

In the previous sections it has been shown that equations of the form

\[(A.10')\] \( Q_1 = N_{m,1} H_1(m) + N_{m,2} H_2(m) + N_{m,3} \)

\[(A.10'')\] \( Q_2 = N_{m,4} H_1(m) + N_{m,5} H_2(m) + N_{m,6} \)

can be derived for each branch. The flow rates \( Q_1 \) and \( Q_2 \) are positive if directed from node I to node J, \( Q_1 \) is at the side of node I, \( Q_2 \) at the side of node J. In the continuity equation for each node:

\[ \sum Q = 0, \] \( Q's \) are positive if directed towards the node under consideration.

So expression \((A.10')\) has to appear in the equation for node I with negative sign, and \((A.10'')\) in the equation for J with positive sign.

Therefore the coefficients \( M_{ij} \) in eq. \((A.1)\) are built up in the following way. First make all \( M_{ij} = 0 \). Then for each \( m \):

\[(A.25)\]

\[ M_{II} = M_{II} - N_{m,1} \]
\[ M_{IJ} = M_{IJ} - N_{m,2} \]
\[ M_{IO} = M_{IO} - N_{m,3} \]
\[ M_{JI} = M_{JI} + N_{m,4} \]
\[ M_{JJ} = M_{JJ} + N_{m,5} \]
\[ M_{Jo} = M_{Jo} + N_{m,6} \]

where \( I = I(m) \) and \( J = J(m) \). From this procedure follows that if nodes \( i \) and \( j \) are not directly connected by a branch, \( M_{ij} \) and \( M_{ji} \) remain zero. \( M \) therefore is a sparse matrix and it is wise to use a solution procedure that takes advantage of this property. The elimination method was chosen as method of solution; the elements on the main diagonal of the matrix act as pivots, since these elements are usually larger than the others. This leaves only the order of elimination as the degree of freedom that can be used to
optimize the solution. The elimination strategy can be based on the structure of the matrix; with structure is meant the distribution of non-zero elements in the matrix. This structure does not vary from one time step to another, at least not in the case of water flows. Therefore the elimination strategy is determined once and then used for subsequent time steps. The order of elimination is found in the same way as by Churchill (Ref. 8).

In the computation of concentrations the structure of the matrix is in most cases different. As has been pointed out in the previous section, some of the coefficients $N'$ may be zero, depending on the direction of the flow. For the matrix $M'$ this has the following consequence: if the flow is directed from node $i$ to node $j$, then $M'_{ij} = 0$, and $M'_{ji} \neq 0$. Then $C_j$ has to be calculated after $C_i$. No elimination is necessary, direct calculation of the unknowns is often possible. The structure of the matrix $M'$ can be depicted as a directed graph; for a typical example see fig. A.8. There is an arrow (called are in graph theory (ref. 9)) from node $i$ to node $j$, if $M'_{ji} \neq 0$.

The direction is derived from the direction of the flow, and it indicates the order of the calculation of the unknown C's. The calculation starts at 'sources', nodes with no arcs coming to them, and proceeds until the 'sinks' are reached. As long as no circuits exist in the graph, direct calculation is possible. If circuits do arise (see for instance fig. A.8), we have recourse again to the elimination method. So it appears that a combined procedure for the elimination strategy and for the order of calculation is needed.

This procedure will do the following: eliminate unknowns which cannot be calculated directly; elimination amounts to a change in the graph whereby one or more circuits disappear (see fig. A.9). Some arcs disappear, some new ones are formed. This proceeds so long as is necessary for all circuits to vanish. Then direct calculation is possible. Fig. A.9 illustrates the elimination of $C_i$; this elimination means that coefficient $M'_{ki}$ is made zero. To reach this the $i$-th row is substracted a certain times from the $k$-th row. But then, if
$M'_{ij} \neq 0$, $M'_{kj}$ will become different from zero. In terms of the graph this means that the arc \( i-k \) disappears, and for any \( j \) for which an arc \( j-i \) exists, an arc \( j-k \) will appear.

Fig. A.9. Graphical representation of elimination procedure.

The calculation proceeds faster in the case of the concentrations, but this advantage is compensated somewhat by the fact that the solution strategy has to be determined for every time step again, because the structure of the matrix can change every time step, since it depends on the direction of the flow.

The method derived here has so far only been used for the computation of water flows, but no essential problems are expected in its use for pollution problems.
Appendix B. Accuracy of unsteady flow computations in the FLOWS system.

In practical flow computations there are several sources of inaccuracy, such as the schematization, the lack of necessary data, the inaccuracy of measured data, numerical errors etc. In this chapter only numerical errors will be considered. The unsteady flow equation consists of several terms, each of which contributes to the total numerical error. It appears to be too difficult to find reasonable estimates for the error in the full equation, so usually estimates are given for simplified versions of the unsteady flow equation. Two such versions will be considered in the following. Both of them are linearized. One pertains to waves in which inertia and storage are the dominant effects, the other to waves in which friction and storage dominate. In flood routing the latter case is more relevant, in many problems, like flow in estuaries, both cases have to be considered.

B.1. Waves with dominant inertia

The equations for a linearized wave with dominant inertia and storage are

\[
\frac{dQ}{dx} + b \frac{dh}{dt} = 0
\]

(B.1)

\[
gA \frac{dh}{dx} + \frac{dQ}{dt} = 0
\]

(B.2)

Since the equations have been made linear, one can consider the harmonic solutions separately. So suppose that

\[
Q = \bar{Q} e^{i\omega t - ikx} = \bar{Q} \hat{r} \hat{s} \hat{x}
\]

(B.3)

\[
h = \bar{h} e^{i\omega t - ikx} = \bar{h} \hat{r} \hat{s} \hat{x}
\]

(B.4)
where \( \tilde{x} = x/\lambda \) and \( \tilde{t} = t/\tau \), \( \lambda \) and \( \tau \) being the spatial and time steps, respectively. \( \tilde{Q} \) and \( \tilde{h} \) are complex numbers containing both the amplitude and the phase of the wave, and

\[
\begin{align*}
  r &= e^{i\omega \tau} \\
  s &= e^{-i k \lambda}
\end{align*}
\]

If the numerical approximations (A.5), (A.6), (A.7) are substituted into eqs. (B.1) and (B.2), one obtains

\[
\begin{align*}
  \frac{\bar{Q}}{\lambda} (s - 1) (\theta r + 1 - \theta) + \frac{b \bar{h}}{\tau} (r - 1) \frac{s + 1}{2} &= 0 \\
  \frac{g\bar{h}}{\lambda} (s - 1) (\theta r + 1 - \theta) + \frac{c}{\tau} (r - 1) \frac{s + 1}{2} &= 0.
\end{align*}
\]

This set of equations has a non-trivial solution if

\[
(B.5) \quad \frac{gA}{\lambda^2} (s - 1)^2 (\theta r + 1 - \theta)^2 - \frac{b}{\tau^2} (r - 1)^2 \left( \frac{s + 1}{2} \right)^2 = 0
\]

By the introduction of the dimensionless parameter

\[
\mu = \frac{i \sqrt{gA/b}}{\lambda} = C_n \tau / \lambda
\]

relating the step size in space and time, this equation simplifies into

\[
(B.6) \quad 2\mu \frac{s - 1}{s + 1} = -\frac{r - 1}{\theta r + 1 - \theta},
\]

and a second equation which is not interesting.

For waves harmonic in space with wave length \( L \)

\[
s = e^{-ik\lambda} = e^{-2\pi i l/L}
\]

With each value of \( s \) a value of \( r \) is found.
If $\theta > \frac{1}{2}$ one will find values of $r$, for which $|r| < 1$, so there is some numerical damping. The amount of damping per time step is $\log |r|$, so per period it is:

\[(B.7) \quad \alpha = \frac{T}{\tau} \log |r|\]

The analytical model (B.1) and (B.2) does not show any wave damping; so $\alpha$ must be kept low. Fig. B.1 shows a plot of $\alpha$ as a function of $T/\tau$ and $\mu$.

The ratio of the wave velocity in the numerical model and the one in the analytical model, which is $c_w$, is given by

$$n_c = \frac{\text{arg}(\psi)}{c_w \cdot k \cdot \tau} = \frac{T}{2\pi} \text{arg}(r).$$

The value of $n_c$ as a function of $T/\tau$ and $\mu$, is presented in fig. B.1, together with $\alpha$.

The parameter $\theta$ in both graphs is $\theta = 0.55$, being a value often used in practice. No separate graphs for $\theta = 0.5$ are presented, since for $\theta = 0.5$ $\alpha$ vanishes identically, and $n_c$ is the same as for $\theta = 0.55$. 
Fig. B.1. Wave celerity coefficient and numerical damping of a wave with dominant inertia ($\theta = 0.55$).
B.2. Waves with dominant friction

In the case of flood waves the acceleration terms usually are small compared with the friction term. So the following set of equations will serve as starting point for the analysis:

\[ \frac{\partial Q}{\partial x} + b \frac{\partial h}{\partial t} = 0 \]  
(B.9)

\[ gA(\frac{\partial h}{\partial x} + J_o(Q, h)) = 0 \]  
(B.10)

For the purpose of the accuracy analysis it is assumed that \( Q \) is equal to a constant part plus a comparatively small variable part, \( \text{i.e.} \)

\[ Q = \bar{Q} + q \]  
(B.11)

where \( \bar{Q} \) is constant and positive, and \( q \ll \bar{Q} \).

It follows that the depth \( d = h - z \), also can be split

\[ d = d_o + y \]  
(B.12)

where \( d_o \) is constant, and \( y \ll d_o \).

Then (B.9) and (B.10) have to be linearized.

This yields:

\[ \frac{\partial q}{\partial x} + b \frac{\partial y}{\partial t} = 0 \]  
(B.13)

\[ \frac{\partial y}{\partial x} + \frac{\partial z}{\partial x} + J(\bar{Q}, d_o) + J_q q + J_d y = 0 \]  
(B.14)

\[ J(Q, d) = J_o(Q, z + d) \]

\[ J_q = \frac{\partial J}{\partial Q} \]

\[ J_d = \frac{\partial J}{\partial d} \]  
(B.15)
For the steady state holds

\[(B.16) \quad \frac{\partial z}{\partial x} + J(\tilde{q}, d_o) = 0\]

so that

\[(B.17) \quad \frac{\partial y}{\partial x} + J_q q + J_d y = 0\]

By a substitution similar to (B.3) and (B.4)

\[(B.18) \quad q = q e^{i\omega t - ikx} = q r_s x\]
\[(B.19) \quad y = y e^{i\omega t - ikx} = y r_s x\]

into the numerical model one obtains

\[(B.20) \quad Dr \frac{s-1}{\lambda} \tilde{q} + (1 - \theta) \frac{s-1}{\lambda} \tilde{q} + b \frac{s+1}{2} \frac{r - 1}{\tau} \tilde{y} = 0\]
\[(B.21) \quad (\theta r + (1-\theta)) \left( \frac{s-1}{\lambda} y + J_q \frac{s+1}{2} \tilde{q} + J_d \frac{s+1}{2} \tilde{y} \right) = 0\]

Again the determinant should vanish, so

\[(B.22) \quad (\theta r + 1 - \theta) \star \]
\[\{ (\theta r + 1 - \theta) \frac{s-1}{\lambda} \star (\frac{s-1}{\lambda} + J_d \frac{s+1}{2}) - b \left( \frac{s+1}{2} \right)^2 \frac{r+1}{\tau} J_q \} = 0\]

\[(B.23) \quad \text{Obviously one solution is: } r = (\theta - 1)/\theta. \text{ This solution does not lead to any difficulties if } \theta > \frac{1}{2}.\]

In order to simplify the expressions some new parameters are introduced.

\[(B.24) \quad \nu = - \frac{J_d}{bJ} \star \frac{1}{\lambda} = c_f \frac{1}{\lambda}\]
\[\rho = - \frac{1}{Jd} L, \quad \rho' = \frac{L}{\rho} \rho\]
\( c_f \) is the flood wave celerity, and \( \rho \) is a dimensionless number relating depth, slope and wave length.

Then (B.22) transforms into

\[
(\theta r + 1 - \theta) (s - 1) \left\{ -\rho' (s - 1) + \frac{s+1}{2} \right\} + \frac{1}{\nu} \left( \frac{s+1}{2} \right)^2 (r - 1) = 0
\]

\( r \) can now be found without any difficulty

\[
(B.27) \quad r = \frac{\left( \frac{s+1}{2} \right)^2}{\left( \frac{s+1}{2} \right) + \theta \nu (s - 1) \left\{ -\rho' (s - 1) + \frac{s+1}{2} \right\}}
\]

The value of \( r \) resulting from the study of the numerical model is to be compared with the value from the analytical model; from (B.13) and (B.17) results

\[
(B.28) \quad \omega = -\frac{J_d}{b J_q} k + i \frac{k^2}{b J_q}
\]

\[
\omega t = v k l + i v \rho k^2 L
\]

The following two parameters give an impression of the accuracy of the numerical model as a flood routing model:

\[
(B.29) \quad n_c = \frac{\text{arg}(r)}{v k l}
\]

\[
(B.30) \quad n_d = \frac{-\text{Im}(r)}{v \rho k^2 L}
\]

In figs. B.2, B.3, and B.4 \( n_c \) and \( n_d \) have been plotted as functions of \( T/\tau \), with \( v \) (see eq. B.24) and \( \rho \) (see eq. B.25) as parameters. In all these graphs \( \theta = 0.55 \).

The parameter \( \rho \) is a ratio between the characteristic length of the wave and a length inherent to the river, \( d_o / I \). So \( \rho \) is a physical parameter not influenced by the discretization procedure. The parameter \( v \) relates the propagation velocity, time step and distance
step, \( v = c_s \tau / l \).

So for a fixed \( \tau \), \( l \) will increase as \( v \) decreases. It is understandable that generally curves with smaller \( v \) will deviate more from the line \( n = 1 \).

A few tests have been done to check these curves. A numerical model of a prismatic channel has been made. Its dimensions were: \( w = 1 \), \( d_0 = 1 \), \( I = 10^{-4} \), \( g = 10 \). The friction law was of the Chézy type: \( J = 0.004 Q^2 / (g w^2 d^3) \). At the upstream boundary \( Q \) was given as a function of time, its graph is shown in fig. B.5.

The wave period \( T \) can be estimated as \( 400 \times 10^3 \). This gives a value for \( \rho \) of roughly 0.02. The flood wave that would propagate along this channel, was computed with different combinations of step sizes. Good results were obtained with: \( l = 5000 \), \( \tau = 5000 \) (or \( T/\tau = 80 \), \( v = 1 \)) and with \( l = 10000 \), \( \tau = 2000 \) (or \( T/\tau = 200 \), \( v = 0.2 \)). Somewhat less accurate results were found with \( l = 5000 \), \( \tau = 10000 \) (or \( T/\tau = 40 \), \( v = 2 \)).

With still lower \( T/\tau \) unacceptable results were found, e.g. \( l = 10000 \), \( \tau = 20000 \) (or \( T/\tau = 20 \), \( v = 2 \)). An attempt to do a computation with \( l = 10000 \), \( \tau = 40000 \), failed due to divergence of the results. With the exception of the last these results are shown in fig. B.6, in which the discharge in a station at a distance \( 110 \times 10^3 \) from the upstream boundary is plotted.

It is concluded that graphs B.2, B.3 and B.4 are a good instrument to get a first guess of acceptable step sizes. In general with \( v = 1 \) an economical choice is made, as is confirmed by Price (ref. 10).
Fig. B.2. Plots of $n_c$ and $n_d$; $\theta = 0.55$, $\rho = 0.01$. 
Fig. B.8. Plots of $n_c$ and $n_d$; $\theta = 0.55$, $\rho = 0.1$. 
Fig. B.4. Plots of $n_c$ and $n_d$; $\theta = 0.55$, $\rho = 1$. 
Fig. B.5. Flood Routing Problem: Upstream Boundary condition.

Fig. B.6. Flood Routing Problem; discharge at station $x = 110 \times 10^3$. 
Appendix C. FLOWS User's Guide

Contents of this guide

1. System of units
2. Available commands
3. Order of the commands
4. Command descriptions
5. Example of a problem treated by FLOWS
1. System of Units

FLOWS works by default with a set of units derived from the S.I. units m (meter), kg (as unit of mass) and s (second). For instance the standard value of the gravitational acceleration is 9.81 (m/s^2). Other unit systems can be used; the rules follow. Different units can be used for length, width and depth. In the general unit system the units of length, width, depth, time and concentration can be chosen arbitrarily, and the other units follow from the table shown below. In the standard system the unit of concentration is not specified, because it depends on the physical nature of the pollution under consideration. It is recommended that the unit system is chosen such that computed values for the different variables fall in the range between 1 and 100.

<table>
<thead>
<tr>
<th>phys. quantity</th>
<th>standard</th>
<th>general</th>
</tr>
</thead>
<tbody>
<tr>
<td>length, hor. coordinate</td>
<td>1000 m</td>
<td>n1</td>
</tr>
<tr>
<td>width</td>
<td>100 m</td>
<td>nw</td>
</tr>
<tr>
<td>depth, water level</td>
<td>1 m</td>
<td>nh</td>
</tr>
<tr>
<td>time</td>
<td>1000 s</td>
<td>nt</td>
</tr>
<tr>
<td>velocity</td>
<td>1 m/s</td>
<td>n1/nt</td>
</tr>
<tr>
<td>grav. acceleration</td>
<td>1 m/s^2</td>
<td>(n1<strong>2)/nh*nt</strong>2</td>
</tr>
<tr>
<td>friction coeffic. f (=g/C**2)</td>
<td>0.001</td>
<td>nh/n1</td>
</tr>
<tr>
<td>wind factor</td>
<td>0.001</td>
<td>nh/n1</td>
</tr>
<tr>
<td>discharge</td>
<td>100 m^3/s</td>
<td>nh<em>nw</em>n1/nt</td>
</tr>
<tr>
<td>concentration</td>
<td>nc</td>
<td>nc</td>
</tr>
<tr>
<td>transport</td>
<td>nc*100 m^3/s</td>
<td>nc<em>nh</em>nw*n1/nt</td>
</tr>
</tbody>
</table>
2. Available commands
-----------------------

The following user commands are presently available in the FLOWS package.

commands describing the network:

SET set some general parameters
FR friction law
LEV standard levels
CAN canal stretch
PIPE pipe stretch
RIV river stretch
RES reservoir
WEIR weir
CULV culvert
OUTFLOW given stage-discharge relation
PUMP pump
WIND wind data
COOR coordinates of the nodes of the network
DIF diffusion coefficient for transport computation
COU couple nodes or branches (transport computation)
UNCO uncouples nodes or branches (transport computation)
NODE node with given water level and/or concentration
BR branch with given discharge and/or transport

initial values:

INIT
output requests:

PUT copy the complete state of a certain moment onto secondary storage
SKE plot of selected variables via printer
WRI write values of selected variables on disk, card or paper output

computation command:

COMP starts a computation

The commands are described in more detail in section 4. They are given there in alphabetical order.
3. Order of the commands

The commands for a FLOWS job can be given in arbitrary order, unless otherwise specified in the description of the commands. An order which is often useful, is the following:

FLows
SET
commands describing the network
initial values (command INIT)
output requests
COMPUTE
SET (changes in parameters)
commands describing changes in the network repeated
new output requests optional
COMPUTE

Remarks:
A computation obviously changes the values of water levels, discharges, concentrations etc. As the above list shows successive computations may be done. Any computation starts with the values as they were at the end of the previous one, unless the user has applied modifications through the use of the command INIT. Any other parameter is changed only by giving the command that modifies such a parameter, unless otherwise stated.
4. Command descriptions

Language conventions

The FLOWS commands are in the form of a problem oriented language, as for other ICES subsystems. The conventions for the description of the FLOWS commands conform to the STRUDL language conventions (ref. LOGCHER et.al., chapter II). To summarize:

- input is free format
- end of card is end of command, unless a hyphen (–) appears
- in the text 'ABCDEF' only the underlined part 'ABC' is significant.
- a part between parentheses: (.....), is optional
- a choice exists between elements enclosed in braces:

{......}

- if an alternative is indicated by an arrow ->, this choice is default, i.e. it is assumed if none of the alternatives is mentioned by the user
- labels for data items enclosed in parentheses, (.....), may be omitted if the data items come in exactly the same order as in the command description.

In addition a special type of brackets, <......>, is used to indicate a repeat group; such a group of data items should appear at least once, unless otherwise stated.
Purpose:

to introduce a branch in the network, in which the discharge of water and/or the transport of pollutant is given as a function of time.

Syntax:

```
DELETE

BRANCH kb

(FROM kpl) (TO kp2) /

FLOW /A/ (TRANSPO

-> CONDITIONAL

UNCOND

/ )

```

In this scheme the symbol /A/ stands for:

```
-> FIXED uu

PERIOD per COEF a0,b1,a1,b2,a2,....

(DATASET ida

(List ird INIT u0)

VALUES u

```

Function:

DELETE indicates that the branch is to be deleted from the network.

FROM/TO the branch with number kb connects nodes numbered kpl and kp2. The positive direction is from kpl and kp2.

FLOWS means that the discharge is given.

TRANSPORT means that the transport is given. This transport can be given UNCONDITIONALLY, i.e. regardless of the direction of the water flow, or CONDITIONALLY, which means that if the flow is from a node of the network, the transport is equal to the discharge multiplied by the concentration in the node mentioned.
If neither FLOWS nor TRA is given, it is assumed that the data pertain to water flows.
The block indicated by the symbol /A/ contains data concerning the boundary value, it is described in more detail in the command NODE.
This element can be used to describe a boundary condition. In that case usually the part FROM kpl or TO kp2 is absent. It is then assumed that that node is outside the model.
Purpose:
to introduce a (part of a) canal into the network

Syntax:

\[
\text{CANAL \( kb \) (FROM \( kpl \)) \( (TO \) \( kp2 \)) LENTH \( l \) \( (WIDTH \) \( b \) \(- \)
\begin{bmatrix}
\text{DEPTH} \\
\text{D1}
\end{bmatrix}
\)
\( d1 \)(D2 d2)(WIND \( fw \))(FRICITION \( f \))(ROUGHNESS \( r \))
\]

Function:
The branch numbered \( kb \) connects the nodes numbered \( kpl \) and \( kp2 \).
Default values are \( kpl = kb \), and \( kp2 = kpl + 1 \).
The LENGTH of the canal is \( l \).
The WIDTH is \( b \). Default value: \( b \) is equal to the width of the CANAL given on the immediately preceding card. If \( b \) is not given and the preceding command does not pertain to a Canal, the results are unpredictable.
The DEPTH at the side of node \( kpl \) is \( d1 \), at the side of node \( kp2 \) is \( d2 \). Default values: \( d1 \) is equal to the \( d2 \) of the CANAL given on the immediately preceding card, \( d2 = d1 \).
\( fw \) is the reduction factor in the formula for wind stress on the water surface; default: \( fw = 1 \). (see also command WIND).
\( f \) and \( r \) are the values to be used in the friction formula given by the command FR. Default values are given by the same command. The command FR must appear at least once before a CANAL is given.
If $h$ is the water elevation above datum, the actual depth is $h + d$. 
Purpose:

to assign values to the coordinates of the nodes of the network

Syntax:

COORDINATES < (NODE) kp, (X) x, (Y) y >

Function:

x and y are the coordinates of the node kp.
Purpose:
this command starts a computation

Syntax:

\[
\text{COMPUTE} \begin{cases} 
\rightarrow \text{UNSTEADY} /B/ \quad \text{OVER } i1 \ (\text{STEPS}) \ (\text{ITER } i2) \\
\rightarrow \text{PERIODIC} \ (\text{FLO}) \ \text{OV } i1 \ (\text{PER}) \ (\text{AC } r1) \ (\text{IT } i2) \\
\rightarrow \text{STEADY} /B/ \quad \text{ACCURACY } r1 \ (\text{ITER } i2) \\
\end{cases}
\]

Here the block /B/ stands for:

\[
\begin{cases} 
\rightarrow \text{FLOWS (GIVEN)} \\
\rightarrow \text{PERIODIC} \\
\rightarrow \text{UNSTEADY} \\
\end{cases}
\]

Function:

One can choose to compute flows, or concentrations, or both flows and concentrations, steady as well as unsteady.

11 (UNS case) the number of time steps over which the computation should extend.
(Per case) the number of periods over which the computation should extend.
12 (UNS and PER case) the number of iterations per time step (default value: 0).
(STEADY case) the maximum number of iterations (default
value: 10).

rl is the required accuracy. The computation ends when the differences between the values found in two subsequent iterations, are all below rl. If the required accuracy is not reached after i2 iterations, the computation also ends, but a message is printed.

If only concentrations have to be computed, one can specify that the flows are steady, unsteady or periodic; if they are unsteady, they must have been retained in a preceding computation (see command RETAIN).

If one wants to compute periodic flows, the number of time steps per period must be known to the subsystem. The user must supply the command SET PER .... previously. The PERIODIC computation can only be used for flows, not for concentrations.
CoupL@

Purpose:

Couples nodes or branches for transport computation.

Syntax:

\[
\begin{align*}
\text{COUPLE} & \left\{ \begin{array}{c}
\rightarrow \text{BRANCHES} < i 1 \\
\rightarrow \text{BOTH}
\end{array} \right\} > \\
\end{align*}
\]

Function:

The commands COUPLE and UNCOUPLE are related to pollution computations, and in particular to canal, river or pipe branches.

A branch is coupled at the FRONT side if always holds \( c_1 = C(I) \), and uncoupled if this is only the case during inflow from node \( I \). The branch is coupled at the END side if always \( c_2 = C(J) \).

In practice the branches need only be uncoupled at true branching points, i.e. nodes where more than two branches meet, or at nodes where a boundary condition is prescribed. The command COUPLE SIMPLE NODES is intended to couple all other nodes.

If neither command is given, all branches will be uncoupled.
Purpose:
introduce a culvert or similar constriction into the model.
Syntax:

```
CULVERT kb FROM kpl -
```

```
TO kp2
  { -> OPEN (BOTH) (DIR) }
  { CLOSED (NEG) (DIR) }
  \{ FIXED (APERTURE) aa \\
    \{ VARIABLE (AP) <yy,aa> \}
  \} (FAC rf)
```

Function:
The branch number of the culvert is kb. It connects the nodes kpl and kp2.
The culvert can be OPEN for flow in both directions, or CLOSED for flow in the negative direction (from kp2 to kpl). The aperture of the culvert can be constant (FIX) or variable according to the waterlevel in node kpl. In this case aa is the area of the aperture, and yy is the corresponding water level in node kpl (=h1). Several pairs of yy,aa can be given. The values of yy should be given in ascending order. Between these values linear interpolation will be used. If the water level is below the lowest value of yy, the aperture corresponding to this lowest value will be taken. If the level is above the highest yy, the aperture area of the latter will be taken. rf is the discharge reduction factor, default value: rf=1.
The following formula is used for the calculation of the discharge Q:

```
Q = rf*aa*sqrt(2*g*(h1-h2))  if h1>h2
```
Purpose:
To give the diffusion coefficients

Syntax:

\[
\text{DIFFUSION (FIXED } d0) \text{ (COEFFICIENT } dl)\]

Function

The diffusion coefficient will be calculated by means of the formula:

\[
D = d0 + dl*|u|h .
\]

Default values: \( d0 = 0. \)
\( dl = 0. \)

The diffusion coefficient only pertains to branches of the canal, pipe or river type.

The unit of \( D \) and \( d0 \) is in the general unit system (see section 1) \( \text{nl}^2/\text{nh}, \) and in the standard system \( 1000 . \text{m}^2/\text{s} \). The unit of \( dl \) is consequently \( \text{nl}/\text{nh}, \) and \( 1000 \) in the standard system.
FRICTION

Purpose:
to indicate the type of friction law that is to be used in the branches given by the commands after this one

Syntax:

\[
\begin{align*}
\text{FRITION} & \quad \rightarrow \text{CHE } f \\
& \quad \text{POWER } p \quad \text{FACTOR } f \quad \text{ROUGHNESS } r \\
& \quad \text{LOGARITHMIC } \text{FACTOR } f \quad \text{ROUGHNESS } r
\end{align*}
\]

Function:
The shear stress on the water body (S) due to bottom friction can in general be expressed as

\[ S = -f' \cdot |v| \cdot v \]

The friction coefficient \( f' = g/(C^{**2}) \), where C is the usual Chezy coefficient.

In the CHE type \( f' = \) constant \( = f \). The value of \( f \) must be given.
In the POWER law type \( f' = f \cdot (r/h/r)^{**(-2*p)} \). Default values are:
\( p = 1/6 \), \( f = 0.2279 \). The value of \( r \) must be given.
In the LOGARITHMIC type \( f' = f/(\ln(3.2 \cdot r/h/r))^{**2} \).
Default value: \( f = 0.1636 \). The roughness size \( r \) must be given.

\( rh \) means the hydraulic radius.
INIT

Purpose:
initialises variables

Syntax:

```
INIT

\[ \begin{array}{c}
H \\
C \\
Q \\
S \\
\end{array} \right\} \begin{array}{c}
\{ \text{IDENT} \} \\
\text{EQUAL} \ r_1 \ (r_2) \} \\
\text{il} \ (\text{UNTIL} \ i_2) \} \\
\end{array} \right\} \begin{array}{c}
\text{FLOWS} \\
\text{FLOWS} \ (\text{AND}) \ \text{POLUTIONS} \} \\
\text{POLUTIONS} \\
\end{array} \right\} \begin{array}{c}
\text{FROM} \ i_3 \\
\end{array} \right\}
```

Function:
By default all initial values of H (water levels), C (concentrations), Q (discharges) and S (transports) are 0. These values can be changed in one of the following ways:
- giving the values in this command; if all values are the same ID can be used, otherwise the nodes or branches have to be mentioned by their numbers. In the latter case an error message is printed, if an initial value is given for a node or branch which is non-existent at that stage, or in which a boundary condition is given. In the case ID this message does not appear, and the value in the node or branch is left unchanged.
- reading the values for the whole network from a dataset, characterized by the number i3. The same number should appear instead of the periods after FT in the following job control statement:
\[
//FT.F001 DD DSN=...,DISP=OLD,UNIT=DISK,VOL=SER=DISK...
\]
This can be done either for the data pertaining to the water flow, or for pollution data, or both. The command PUT is used to put the state at the end of a former computation onto secondary storage. This facility is used of one wants to use a state computed before, as initial state for the present computation.

The command INIT should follow the commands describing the network.

\[
\begin{align*}
&i_1 = \text{node or branch number} \\
&i_2 = \text{node or branch number} \\
&r_1 = \text{value for } H, C, Q_1, \text{ or } S_1. \\
&r_2 = \text{value for } Q_2 \text{ or } S_2; \text{ if absent, the program takes } Q_2 = Q_1 \\
&\text{or } S_2 = S_1, \text{ as the case may be.}
\end{align*}
\]
LEVELS

Purpose:

to give a set of standard levels.

Syntax:

LEVELS y_1, y_2, y_3, ...

Function:

The standard levels $y_1, y_2, y_3, ...$ can be used if some quantity is given as a function of the water level. See for instance the command RIV.
NODE

Purpose:
to introduce a node where water level and/or concentration are given as a function of time.

Syntax:

\[
\text{\{ WATER \}} \\
\text{\{ LEVEL \}} /A/ \ ( \text{CONCENTRATION} /A/ ) \\
\text{\{ NODE \} \{ kp \} \{ DELETE \}}
\]

In this scheme the symbol /A/ stands for:

\[
\text{\{ \rightarrow FIXED uu \}} \\
\text{\{ PERIOD per COEF a0,bl,a1,b2,a2,..... \}} \\
\text{\{ (LIST ird INIT uu0) \}} \quad \text{(LIST \ ird \ INIT \ uu0) \ \text{VALUES <u>}} \quad \text{(LIST \ ird \ INIT \ uu0) \ \text{VALUES <u>}} \\
\]

Function:
kp is the number of the node.
DELETE means that the node is deleted from the network.
WATER or LEVEL means that data follow which give the water level in the node. If no data for the concentration follow, both words may be left out.
CONC means that data for the concentration in the node follow.
FIXED means that the value is fixed, equal uu.
PERIOD: the value is a periodic function of time, given by means of the Fourier coefficients, which follow after COEF.
The meaning of the Fourier coefficients is as follows:
\[
f(t) = a0 + \text{Sum} \ (b(i) \sin(2\pi i t/\text{per}) + a(i) \cos(2\pi i t/\text{per}) )
\]
LIST: a list of values is given, the interval between two
values is ird time steps. If 'LIST ird' is left out, ird is set equal to 1.
The value at time 0 is given by INIT u0.
DATASET ida: the list of values is found on dataset nr. ida.
In the job control statements a description of the dataset must be given in the form:
//F001 DD DSN=...,UNIT=...,VOL=SER=...,DISP=OLD
The number ida must appear on the two dots in the name F001.
VALUES u1,u2,u3,...: here u1,... are the values themselves,
u is the value of the function at time: i * ird * dt.
If during the computation the end of the list is reached, the program returns to u1. Thus the list can also be used for a periodic function.
OUTFLOW

Purpose:
to give a stage-discharge relation.

Syntax:

OUTFLOW i1 FROM i2 -

0

\[
\text{(TO i3) } \begin{cases}
\text{-> DISCHARGE } \langle r1, r2 \rangle \\
\text{RIVER SLOPE } r1 \text{ WIDTH } r2 \text{ DEPTH } r3 \text{ FRICTION } r4
\end{cases}
\]

Function:
The branch number of the outflow element is i1. It connects the nodes i2 and i3. Node i3 is optional.
There are two versions of the outflow element. In the case DIS r1 is a water level in node i2, and r2 the corresponding discharge in the branch i1. For other rules see command CULV. In the case RIV the stage-discharge relation is the same as in a river with uniform flow. The slope, width, and depth of the river have to be given. The friction factor r4 is equal to \( g/(C^2) \), where C is Chezy's coefficient.
The element can be used as boundary element. In that case the node i3 usually is absent.
Purpose:
To introduce a pipe or pipe stretch into the network.

Syntax:

\[
\text{PIPE } \text{kb FROM } \text{kp1 TO } \text{kp2 LENGTH } l \text{ DIAMETER } d \text{ ELASTICITY } e -
\]

\[
\{ \rightarrow \text{FULL} \}
\]

\[
\{ \text{CAVIT} (\text{LEVEL}) \ hcl \ (\text{END} \ hc2) \} \quad \text{FRICITION } f \text{ ROUGHNESS } r
\]

Function:
For kb, kp1, kp2, l, f, and r see command CANAL.
\[
d \quad \text{diameter of the pipe. Default: equal to the diameter of the pipe on the immediately preceding card.}
\]
\[
e \quad \text{storage coefficient. It combines the storage by compression of the water and by elasticity of the pipe.}
\]
\[
e \quad \text{is increase of the water mass in a unit length of the pipe} / \text{ (increase of the piezometric head} \ast \text{ density of the fluid). Default as with } d. \text{ e may be equal to } 0, \text{ if the compressibility effect is negligible.}
\]

\[
\text{FULL means that no cavitation will occur.}
\]
\[
\text{CAV means that cavitation will occur in section 1 as soon as h1<hc1, and in section 2 as soon as h2<hc2.}
\]
\[
\text{Default: hc2=hc1}
\]
Purpose:
to introduce a pump into the network.

Syntax:

\[
\begin{array}{c}
PUMP \text{ i1 FROM i2 TO i3 (DISCHARGE) <r1, r2>}
\end{array}
\]

Function:
The discharge in branch i1 depends in a way given by the user on the difference in water level in the nodes i1 and i3. \( r_1 \) is a difference in head between nodes i2 and i3, and \( r_2 \) the corresponding discharge. If \( H(i2)-H(i3)=r_1 \), then \( Q(i1)=r_2 \). The values of \( r_1 \) should be given in ascending order. If the difference in head is lower than the smallest value of \( r_1 \), the discharge corresponding to this \( r_1 \) will be taken. If it is higher than the largest \( r_1 \) (largest and smallest in the algebraic sense), the discharge corresponding to this \( r_1 \) will be taken. Otherwise linear interpolation will take place. See the figure below.
Purpose:

To produce a copy of the state of a certain moment onto secondary storage

Syntax:

\[
\begin{align*}
\text{PUT} & \quad \{ \ \rightarrow \text{FLOW} \} \\
& \quad \{ \text{FLOW} \ (\text{AND}) \ \text{POLLUTION} \ \text{ON} \ il \} \\
& \quad \{ \ \text{POLLUTION} \} \\
\end{align*}
\]

Function:

The present state of the network is written on secondary storage; so, if one wants to retain the state at the end of a certain computation, one gives a PUT command immediately after the COMPUTE command.

il is the datasetnumber appearing in the Job Control statement

//FT..F001 DD ....

The data written by the command PUT onto secondary storage, can be used as initial state for a subsequent computation. The command INIT is used for this purpose.
Purpose:
an aid in debugging the user's input or the subsystem itself

Syntax:

```
REPORT

COORDINATES

NETWORK (REPAIR)

REAL

INTEGER

-> MIXED

COMMON (k1 (UNTIL k2))

WBDA
WPDA
WGDA
NETW
FLO
TFLO
H
COEF
KTERW
KTERC
CONS
CBDA
CPDA
CGDA
C
```

Function:

REPORT COORDINATES checks:
1. whether coordinates of nodes have been given at all,
2. whether coordinates of any node are zero; the user may have forgotten to give the coordinates of this node. Such nodes are reported by their numbers.
REPORT NETWORK checks:
1. whether the nodes connected by each branch are of non-zero type (zero means non-existent); if a node of zero type is encountered, and if REPAIR is requested, the type is made 1, i.e. a normal internal node.
2. whether the network is simply-connected; if an isolated node is encountered, it is reported, and if REPAIR is requested, its type is made 0 (non-existent). If the network consists of more than one disconnected components, each component is reported by means of a list of the nodes contained in this component.

Both other facilities require a more detailed knowledge of the working of the subsystem. Both serve to find out whether data have been stored correctly, and/or to show intermediate results of the computation. The subsystem documentation provides information on the contents of the subsystem Common and the arrays of the subsystem.

REPORT COMMON prints the k1-th until the k2-th full word of the common area, either in real format, or in integer format, or both. By default it prints the complete subsystem common.

REPORT ARRAY ... prints the contents of the mentioned dynamic array.
RESERVOIR

Purpose:
to add a reservoir to the model, optionally combined with a
weir, culvert etc.

Syntax:

```
RESERVOIR i1 AT i2 {
    -> UNIFORM (SURFACE) r1 (DEPTH r2)
    VARYING (SURFACE) < r1, r2 >
}
```

(WITH) {
    WEIR
    CULVERT
    OUTFLOW
    PUMP
}

Function
The reservoir acts as a pure storage element. Together with
the weir etc. it can form one branch, with branch number i1.
It is attached to node i2, and if a weir etc. is present, it
may connect two nodes, i2 and i3. For the blocks W, C, O, or
P, see the commands WEIR, CULV, OUTF or PUMP, resp.

In the case UNIFORM the surface of the reservoir is
independent of the level in the reservoir. r1 is this
surface. If concentrations have to be computed, the content
of the reservoir is important; then the depth r2 must also
be given. If r2 is not given it is assumed to be 10. So if
one computes only flows, one does not run into trouble,
unless the level sinks below $-10$.

In the case VARYING the surface depends on the water level. $r1$ and $r2$ are water level and corresponding reservoir surface resp. The surface corresponding to a certain water level is calculated in the same way as described in the command CULV. The water content is also calculated. It is assumed to be $0$ if the water level is below the lowest value of $r1$. Above this it is found by integrating the surface. The levels have to be given in ascending order.

In the composite branch $Q1$ is the flow rate into the reservoir from the node $i2$. $Q2$ is the discharge through the weir etc. In the concentration computation it is assumed that the concentration in the reservoir is equal to that in node $i2$, and that the transport through the weir etc. is equal to $Q2$ times the concentration at the upstream side.
RETAIN

Purpose:
Retain flow data for a transport computation.

Syntax:

\[
\begin{align*}
\text{RETAIN} & \quad \rightarrow \quad \text{FLOWS} \\
\text{OFF} & \\
\end{align*}
\]

Function:
The command RETAIN FLOWS causes the subsystem to retain in the computer memory the flow data that will be calculated during the unsteady flow computations that follow. These values will be kept in memory only during one job. See commands WRITE or PUT for retaining data for use in other jobs.
The command RETAIN OFF stops the above action. Flow data that were retained already, are not lost, but new flow data will not be retained any more.

In a periodic flow computation the flow data are automatically retained. Retaining flow data can be necessary for later use in transport computations. The flow data are stored and retrieved using the time step counter as an index. It is important that the time step counter is set to 0 at the beginning of a computation in which flows are retained; if this counter is not 0, it can be made 0 by:

\[
\text{SET INT 0}
\]

If the end of the series of data is encountered during a computation in which retained flow data are used, the retrieving starts again at the beginning of the series. Therefore, if the flow data have been generated during a periodic flow computation, no special care is necessary in order to retrieve the right data.
Purpose:
To introduce a river or river stretch into the network, consisting of one or more elementary branches.

Syntax:

```
RIVER kb (FROM kpl) (TO kpe) { -> LEVELS
  STANDARD { -> EQUAL
  STORAGE } } { -> SHALLOW
  NARROW } (WIND fw)
```

On the next nn lines (records):

```
LEN l (FRICITION f) (ROUGHN r) (AS as) (AB ab)
  PROFILE <(y), bs, (bb), (rh)>
```

on the last line (record) of this command:

```
LAST
```

and then similar data as in the previous lines, except: LEN l

Function:
The river stretch consists of nn elementary branches, the first is numbered kb, the next one kb+1 and so on. Branch kb connect the nodes numbered kpl and kpl+1, branch kb+1 the nodes kpl+1 and kpl+2, and so on until kb+nn-1, which connects kpl+nn-1 and kpe.

Default values are: kpl = kb, kpe = kpl +nn.
The RIVER command must be preceded, not necessarily immediately, by the FR command. The first record gives some general data concerning the river under consideration. The next records give information on the subsequent stretches of
the river. On the i-th record after the first, LEN l gives the length of branch kb+i-1, and the other data pertain to the cross-section in the node kpl+i-1. By means of PR f and RO r new values of and r may be given if these are different from the ones given before. After PROFILE the data for the cross-section must be given. y is a level, it must be given if in the first card LEV is specified, otherwise not. bs is the stream width at that level, or at the j-th standard level. bb is the storage width at the same level, it must be given if in the first card STOR is specified, otherwise not. rh is the hydraulic radius at the same level, it must be given if on the first card NA is specified, otherwise not. as is the area of the stream cross-section under the lowest level y, ab is the total wetted area under the same level. Defaults: as=0, ab=as.

The Profile given on the k-th record of the series of records starting with LEN ..., describes the cross-section in node kpl+k-1. It is at the same time the cross-section at the end of branch kb+k-2 (if k>1), and at the front of branch kb+k-1. The Profile indicated as LAST describes the cross-section at the end of branch kb+nn-1.

Remark: if kpe is not equal to kpl+nn, an node with number kpl+nn will be generated. Such often unwanted nodes can be deleted by the command REPORT NETWORK REPAIR.
SET

Purpose:
assign values to several general parameters.

Syntax:

SET (GRAV g) (STEP dt) (TIME t0) (INT it0) (PERIOD nper) --- ---- ---- -- --- ---

(PRINT ntprt) (NAME 'title') (REL the2) (NOTESt) --- --- ---

(HCHK hchk) (WEIRC wcorr)

Function:

GRAV is used to assign a new value to the gravitational acceleration. Default value: g=9.81
STEP the time step dt. This value must be given at least once before the first COMPUTE command.
TIME the initial time. Default: t0=0.
INT the initial value for the time step counter. Default: it0=0
PER the number of time steps per period. This period is then the default period for all periodic boundary values (see commands NODE and BRANCH).
PRINT: ntprt is the number of time steps between two standard prints. Default: ntprt=1
NAME: 'title' is the title of the problem under consideration. This title is at most 40 characters long, and enclosed in quotes.
REL gives the possibility of introducing a numerical damping. The amount of damping is indicated by the coefficient the2. If the2 is above 1., more weight in the numerical scheme is put on the time t+dt than on time t. This results in numerical damping, which is advantageous if discontinuities in the boundary conditions appear. A value of 1.1 is advisable. Default value: the2=1., i.e. the case without numerical damping
(and theoretically the most accurate). The number $\theta$ is equal to 2 times the well known coefficient $\theta$.

NOTEST prevents some tests which are otherwise performed in the numerical procedure, primarily the test on zerodivides in the computation of the system of linear equations.

HCHK: during the computation of the flows the values of the water levels are checked. It is assumed that it is of no use to continue the computation if one or more of these levels are outside a certain range. Therefore the computation is terminated $\text{abs}(H(i)) > \text{hchk}$. The value(s) of $i$ are reported. The checking procedure is not done if $\text{hchk} < 0$. Default value: hchk=1000.

WEIRC: in the computation of the discharge over a weir a correction is used in order to obtain a more stable scheme. The amount of correction is determined by the factor $\text{wcorr}$. Default: $\text{wcorr}=0.2$
Purpose:

produces a plot of computed quantities by means of the printer.

Syntax:

```
SKETCH (ON) il (EVERY i2 STEPS) -
   (AXIS r1) (MARK i3) (UNIT i4) -
   [H]
   [Q1]
   [Q2]
   [V1]
   [V2]
   [A1]
   [A2]
   [R1]
   [R2]
   [B1]
   [B2]
   [C]
   [S1]
   [S2]
   [C1]
   [C2]
   <
   i5 SYMBOL 'a' (FACTOR r2) (OFFSET r3) >
```

Function:

il dataset number. The same number must also appear in the J.C.L statement:

```
//FT..F001 DD SYSOUT=A,DCB=(LRECL=132,RECFM=F)
```

il can be any positive integer under 20, not equal 5,6,
or 7. More than one plot can be made. These are distinguished by different dataset numbers.

\textit{i2} is the number of time steps corresponding to one line distance; default: \textit{i2}=1.

\textit{rl} relative position of the axis. If \textit{rl}=0, the axis is far left, if \textit{rl}=1, it is far right. Default: \textit{rl}=0.5

\textit{i3} number of lines between two markings (see example). Default: \textit{i3}=20.

\textit{i4} size of unit value of the plotted quantity, measured as number of positions on the paper. default: \textit{i4}=10.

\textit{i5} node or branch number.

Variables that can be plotted are:

\begin{itemize}
\item \textbf{H} water level in node \textit{i5}
\item \textbf{Q} flow rate in branch \textit{i5}
\item \textbf{V} water velocity in branch \textit{i5}
\item \textbf{A} flow profile in branch \textit{i5}
\item \textbf{R} hydraulic radius in branch \textit{i5}
\item \textbf{B} total wetted profile in branch \textit{i5}
\item \textbf{C} Concentration in node \textit{i5}
\item \textbf{S} transport in node \textit{i5}
\item \textbf{C} concentration in branch \textit{i5}
\end{itemize}

\textit{a} is the symbol that will be printed to indicate the value of the variable.

If the variable itself cannot be plotted properly, it can be scaled and shifted:

\textit{r2} is the factor by which the variable will be multiplied (default: \textit{r2}=1.)

\textit{r3} is the real number that will subsequently be added (default: \textit{r3}=0.)

\textbf{-------------------}

\textbf{')} 1 added to the symbol refers to the beginning of branch \textit{i5}, 2 refers to the end of the branch.
If during a certain computation a plot has to be produced, the \textit{SKETCH} command has to be given before the \texttt{COMPUTE} command that starts this computation. An example of the plot is shown on the next page. The time points downward, the value of the variable from left to right; left is negative, right positive.
Figure. Example of a plot produced by means of the command SKETCH. (time is vertically downward)
Purpose:
Uncouples nodes or branches for transport computation.

Syntax:

\[
\begin{align*}
\text{UNCOPUPE} & \quad \{ \\
\rightarrow \text{BRANCHES} & \quad <i_1> \\
\rightarrow \text{END} & \quad \} > \\
\rightarrow \text{BOTH} & \quad \} \\
\text{NODES} & \quad <i_2> \\
\end{align*}
\]

Function:
The commands \texttt{COUPLE} and \texttt{UNCOPUPE} are related to pollution computations, and in particular to canal, river or pipe branches.

A branch is coupled at the \texttt{FRONT} side if always holds \( c_1 = C(I) \), and uncoupled if this is only the case during inflow from node I. The branch is coupled at the \texttt{END} side if always \( c_2 = C(J) \).

In practice the branches need only be uncoupled at true branching points, so nodes where more than two branches meet, or nodes where a boundary condition is prescribed.

If neither command is given, all branches will be uncoupled.
Purpose:
introduce a weir or similar constriction into the model.

Syntax:

\[
\begin{array}{c}
\text{WEIR } \text{kb FROM kpl} \\
\text{(TO kp2)} \quad \{ \text{SUBMERGED} \\
\text{CLOSED (NEG)} \} \\
\text{CRITICAL} \\
\text{CREST cl WIDTH wi (FACTOR rf)}
\end{array}
\]

Function:
FROM / TO : see command BRANCH. In the case SUB both submerged and critical state are allowed. In the case CLOSED the same holds, but flow in the negative direction is not permitted.
In the case CRIT there is always critical flow from node kpl. This version of the element can be used as boundary element. If node kp2 is absent, CRIT is assumed. Otherwise, SUB is assumed.
cl is the crest level above datum.
wi is the width of the weir.
rf is the (constant) discharge reduction factor; default value is rf=1.

The formula used for the calculation of the discharge over the weir is in the submerged case:
\[
Q = rf \times wi \times (h2 - cl) \times \sqrt{2g(h1 - h2)} \quad \text{if } h1 > h2
\]
\[
h1 = h(kpl) \quad \text{,} \quad h2 = h(kp2)
\]
in the case of critical flow over the weir:
\[
Q = rf \times wi \times (2/3) \times (h1 - cl) \times \sqrt{(2/3)g(h1 - cl)}
\]
Purpose:
to give wind stresses on the water surface

Syntax:

\[
\text{WIND ( POWER p ) ( FACTOR f ) } \quad \text{--}
\]

\[
\begin{cases}
\rightarrow \text{STEADY VELOCITY w DIRECTION a} \\
\text{UNSTEADY ist } <(\text{VEL}) \text{ w (DIR) a}> \\
\text{NO}
\end{cases}
\]

Function:
p and f are constants appearing in the formula which relates the wind velocity with the stress on water surface:

\[
\begin{align*}
S_x &= -f \cdot (w^{**}p) \cdot \cos(a) \\
S_y &= -f \cdot (w^{**}p) \cdot \sin(a)
\end{align*}
\]

where \(S_x\) and \(S_y\) are the components of the stress vector divided by the density of the water. The direction of the wind with respect to the \(x\)-axis (the same as used in command COORD), \(a\) is to be given in degrees.

Often used values for \(p\) and \(f\) are:

\[p = 2. \quad \text{and} \quad f = 0.36 \times 10^{**-5}\]

If \(m\) are chosen as unit of depth and \(km\) as unit of length, \(f\) should be multiplied by 1000. Default values are: \(p=2.,\)

\(f=0.0036.\)

\(w\) is the absolute value of the wind velocity. In the UNSTEADY case the first couple of values of \(w\) and \(a\) pertains to time \(0\), and the \(j\)-th couple to time: \((j-1)*\text{ist}^*\text{dt}^*\). If not enough data are given, an error message results.

NO means that there is no wind. If the command WIND does not appear at all, 'no wind' is assumed.

Remark: if there is a wind, the coordinates of all nodes
have to be given. This is done by means of the command
COORD.
WRITE

Purpose:

outputs computed quantities to card punch or disk storage

Syntax:

```
WRITE (ON) i1 (EVERY 12 STEPS) FORMAT
    {  -> CARDS
       DISK
       PAPER
    }

<
  {        }
    H
    Q1
    Q2
    V1
    V2
    A1
    A2
    R1
    R2
    B1
    B2
    C
    S1
    S2
    C1
    C2

<_{15 (UNTIL 16) (FACTOR r2) (OFFSET r3)}>
```
Function:

11 dataset number. This number must appear in the J.C.L. statement, which is according to one of the following forms:

```
//FT..F001 DD SYSOUT=C(LRECL=80,RECFM=F)
    with FORMAT C

//FT..F001 DD DSN=.......
//    DCB=(LRECL=80,RECFM=FB,BLKSIZE=2400)
    with FORMAT C

//FT..F001 DD DSN=.......
//    DCB=(LRECL=4,RECFM=FB,BLKSIZE=2400)
    with FORMAT D

//FT..F001 DD SYSOUT=A
    with FORMAT P
```

i2 is the number of time steps between two outputs; default: i2=1.
i5 and i6 are branch or node numbers.
for H, Q1, Q2, etc. and for r2 and r3, see command SKETCH.
5. Example of a problem treated by FLOWS

One of the possible fields of application of FLOWS is flood control in rivers. It is assumed that we have to do with a river with the cross-section shown below:

![Diagram of a river cross-section](image)

Somewhere along this river a flood prevention structure has to be built. The following scheme is proposed and tested by FLOWS: a canal of 3 km length connects the river (at node 3) and an artificial lake (at node 11). At the entrance of this lake a weir is built over which excess floods should flow into the lake, together with a culvert of a small aperture, through which the lake can empty itself after a passage of the flood wave. The weir is built in such a way that outflow from the lake over the weir is prevented.

![Diagram of a river layout](image)
Several plots are requested by the user's commands. The first shows the inflow into and the outflow from the river stretch, and the inflow into the flood prevention structure, the second shows the situation at the inlet of the lake. The effect of the proposed structure can be shown by a comparison with a computation of the river alone.

The period of computation is split into two parts; during the first part the inflow is varying, during the second part it is constant, and equal to the initial discharge. The initial condition (steady flow) is fully known in this example, otherwise a previous computation of the steady state would be necessary, either by means of the command 'COMPUTE STEADY FLOWS' or by 'COMP UNSTEADY FLOWS' with a steady inflow.

The complete job that served to present this problem to the computer, is shown below.
Part of the plot that results from this job is shown in the figure in the description of the command SKETCH.

complete input for the sample problem:

FLOWS
SET NAME 'FLOOD WAVE PROPAGATION'
FRICITION CHEZY 6.25 $ UNIT: .001
RIVER 1 STOR $ UNIT OF LENGTH: 1 KM, DEPTH: 1 M, WIDTH: 1 M
LENGTH 5. PROFILE 6.,10.,10., 6.6, 10.,11., 7.,12.,40.
LEN 5. PROF 5.,10.,10., 5.6,10.,11., 6.,12.,40.
LEN 5. PROF 4.,10.,10., 4.6,10.,11., 5.,12.,40.
LEN 5. PROF 3.,10.,10., 3.6,10.,11., 4.,12.,40.
LEN 5. PROF 2.,10.,10., 2.6,10.,11., 3.,12.,40.
LEN 5. PROF 1.,10.,10., 1.6,10.,11., 2.,12.,40.
LAST PROF 0.,10.,10., .6,10.,11., 1.,12.,40.
CANAL 10 FROM 3 TO 10 LEN 3., DEPTH -3.8, WIDTH 5.
WEIR 11 FROM 10 TO 11 CLO NEG WIDTH 5., CREST 5.0
RESERVOIR 12 AT 11 UNIFORM 200. WITH CULV TO 10 APERT 0.2
OUTFLOW 9 FROM 7 RIVER SLOPE 0.2 WIDTH 10., DEPTH 0. FR 6.25
BRANCH 8 TO 1 FLOW LIST 2 INIT 2. VALUES
INIT H 1,6.5, 2,5.5,3,4.5, 4,3.5, 5,2.5,
   6,1.5, 7, 0.5,10,4.5,11,4.5,
   Q 1 UNTIL 6 EQ 2.,Q 9 EQ 2.
SET STEP 6. PRINT 4 REL 1.1
SKETCH ON 10 AXIS .1 MARKS 10 Q1 8 SYMBOL 'l' FAC 0.5
   Q1 10 SYMBOL 'c' FACTOR 0.5 Q1 9 SYMBOL 'o' FAC 0.5
SKETCH ON 12 AXIS 0.3 UNIT 20 MARKS 10
   H 10 SYM 'h' OFFSET -5., H 11 SYM 'r' OFFSET -5.,
   Q1 11 SYM 'w', Q2 12 SYM 'c'
WRITE ON 16 FORMAT P
   H 3 A1 3 R1 3 Q1 3 V1 3 B1 3
   H 4 A1 4 R1 4 Q1 4 V1 4 B1 4
COMP 36
BRANCH 8 TO 1 FLO FIX 2.
COMP 30
FINISH