ALEID2000® UK version
User Manual

Nieuwegein, May 2003

Research and Consultancy

Groningenhaven 7
P.O. Box 1072
3420 BB Nieuwegein
The Netherlands
Phone: +31 - 30 606 95 11
Fax: +31 - 30 606 11 65
e-mail: alg@kiwaoa.nl

CONTENTS PAGE

1	INTRODUCTION	1-1
1.1	About the program	1-1
1.2	Technical support	1-1
1.3	Quick start	1-1
2	GENERAL DESCRIPTION OF THE PROGRAM	2-1
2.1	Introduction	2-1
2.2	Hydraulic calculations	2-1
2.3	Water quality calculations	2-1
2.4	Data input	2-2
2.5	Output presentation	2-3
3	INSTALLATION AND START-UP OF ALEID2000	3-1
3.1	System requirements	3-1
3.2	Installation	3-1
3.3	Adjusting settings after the installation	3-2
3.4	Working with ALEID2000	3-2
4	BUILDING MODELS IN ALEID2000	4-1
4.1	Collection of relevant model information	4-1
4.2	Required input data	4-2
4.3	Tools helping in setting up a model	4-2
4.4	Model validation and calibration	4-4
4.5	General remarks about the input files for ALEID2000	4-5
4.6	The project file	4-5
4.7	Model input file 1: Basic file	4-7
4.8	Model input file 2: Demand patterns and factors library	4-8
4.9	Model input file 3: Status and control settings file	4-10
4.10	Model input file 4: Water quality options	4-10
4.11	Model input file 5: Co-ordinates file	4-11
4.12	Model input file 6: Curves library file	4-11
4.13	Initialisation file ALCALC.INI	4-12
4.14	Model input file 7: Contour file	4-12
4.15	Measurements data file (menu option “Graph - Options”)	4-13
5	WORKING WITH THE PROGRAM	5-1
5.1	Menu conventions	5-1
5.2	Overview of the menu options	5-2
5.3	Window types	5-3
5.4	File menu	5-4
5.5	Main menu: option “Edit”	5-10
5.6	Main menu: option “Calculate (F5)”	5-32
5.7	Main menu: option “Info”	5-35
5.8	Main menu: option “View”	5-41
5.9	Main menu: option “Graph”	5-48
5.10	Main menu: option “Report”	5-53
5.11	Main menu: option “Window”	5-55

5.12	Main menu: option “Help”	5-56
6	EXAMPLE	6-1
6.1	Introduction	6-1
7	REFERENCE	7-1
7.1	[CONSUMPTION]	7-1
7.2	[CONTROLS]	7-2
7.3	[COORDINATES]	7-4
7.4	[CORRECTION]	7-5
7.5	[DAYFACTORS]	7-6
7.6	[END]	7-7
7.7	[GENERAL]	7-8
7.8	[HYDRANTS]	7-10
7.9	[LEAKAGE]	7-11
7.10	[OPTIONS]	7-12
7.11	[PATTERNS]	7-14
7.12	[PIPES]	7-15
7.13	[PRESS.DEPENDENCE]	7-16
7.14	[PUMP CURVES] and [HYDRANT CURVES]	7-17
7.15	[PUMPS]	7-19
7.16	[QOPTIONS]	7-20
7.17	[QPATTERNS]	7-21
7.18	[QTIMES]	7-22
7.19	[QUALITY]	7-23
7.20	[REACTIONS]	7-24
7.21	[RELATIVE GROWTH]	7-26
7.22	[REPORT]	7-27
7.23	[SOLUTION]	7-29
7.24	[SOURCES]	7-30
7.25	[TANKS]	7-31
7.26	[TDATA]	7-32
7.27	[TIMES]	7-33
7.28	[TITLE]	7-34
7.29	[VALVES]	7-35

APPENDICES:

APPENDIX A - Composition of the program
APPENDIX B - Error messages
APPENDIX C - Files in the program and working directories
APPENDIX D - Files created in ALEID2000
APPENDIX E - Theoretical background of the calculation
APPENDIX F - Troubleshooting
APPENDIX G - Overview of units in ALEID2000
APPENDIX H - Overview of data in ALEID2000

PREFACE

The program presented in this manual, ALEID2000, is the result of the research project “Development of a computer model for residence times and substance propagation”, executed between 1994 and 1996, and subsequently expanded and updated. The purpose of this project was to make available to the water supply companies in The Netherlands a user-friendly network calculation program that could perform water quality calculations.

The program was originally developed within the Dutch water sector on the basis of the EPANET 1.1c network calculation software, which was made available by the US Environmental Protection Agency (US EPA).

The program, in its Dutch version, is used almost universally by the Dutch water supply sector. This version, Aleid2000 UK version, is the first full version in the English language.

We want to encourage all users to let us know about their experience and potential problems with the program. In the past, this has always been a good indicator of the program’s usability and of users’ wishes, which were taken into grateful consideration when further developing the program.

1 Introduction

1.1 About the program

ALEID2000 is a computerprogram for water distribution network calculations. It can be used both for hydraulic calculations, where pressures and flows are determined, and water quality calculations, where distribution of incoming volumes, concentrations and residence times are considered. The program runs on a PC, under Windows 95 or better.

1.2 Technical support

If you have questions about the installation or use of this program, or problems in setting up a model, please contact the ALEID help desk at Kiwa Research and Consultancy, e-mail address: aleid@kiwa.nl

1.3 Quick start

Extensive information on the use of Aleid2000 is given in this manual. For a "quick start" with the program, start with Chapter 3, where the installation procedure and the first steps in using the program are explained. A hands-on example is given in Chapter 6.

The remaining chapters go into more detail about:

- the most important elements in establishing a new model (Chapter 4);
- input files (Chapter 4);
- menu options (Chapter 5);

The theoretical background of the calculation models used is presented in Appendix E.

All users are advised to go through Chapters 3 and 6, and the sections about input layout in Chapter 4. For those who consider network calculations to be a new topic, we suggest first reading the general information in Chapter 2, along with the beginning of Chapter 4 (theoretical considerations and data collection).

2 General description of the program

2.1 Introduction

ALEID2000 performs water distribution network calculations; both hydraulic calculations and water quality calculations.

The calculations can be executed to solve the whole gamut of practical problems, such as:
- design of network extensions and pumping stations;
- network maintenance;
- assessment of network reliability;
- development of effective flushing programs;
- operation of reservoirs;
- diagnosis of water quality problems;
- operational management, regarding both water quality and pressure.

To perform calculations with ALEID2000, a distribution network model must first be created. This model is formulated as a set of input files, which together form a model project. The input files can be created through menus, or with the help of an editor. The results of the calculations can be presented both in graphical and in tabular form.

2.2 Hydraulic calculations

ALEID2000 performs hydraulic analyses for a given network model – primarily the calculation of heads and flows in the network – for specified time steps. Based on this, also derived parameters, such as pressure and pressure gradients, flow velocities or pumping capacities are determined.

The program calculates a number of steady states in a network: one at a particular moment (static or "snapshot"), or several within a certain period, at time steps defined by the user ("quasi-dynamic" simulation). A coherent input model, based on the water balance of the area, is essential for correct hydraulic calculations.

The theoretical background of the hydraulic calculations is comprehensively discussed in detail in the first part of Appendix E.

2.3 Water quality calculations

Water quality calculations in ALEID2000 are based on hydraulic information, combined, in some cases, with a reaction model. The source of calculation principles here is EPANET, network calculation software developed by the United States Environmental Protection Agency.

The program determines the following water quality parameters:

- average travel times in the network;
- contribution from a supply point: this is represented as a percentage of the total delivery at a certain point in the network;
- distribution of conservative substances: these are substances that do not react during transport, such as a concentration of salt;
- distribution of non-conservative substances: a reaction model is required for these calculations. In this case, the hydraulic model has to be extended with additional information required for the reaction model. The program already contains models for chlorine and trihalomethane. Others may be incorporated upon specific request.

Calculation of other water quality parameters is based on the hydraulic situation. A detailed description of the water quality algorithm is given in Appendix E (section E.8).

2.4 Data input

To calculate useful output, a correct input model is required. Such a model describes the network: pipes, valves, supply and demand based on the water balance and network configuration in the selected area. The network is then schematised into a system of nodes and links (lines).

Possible node types are:
- demand node: various demand patterns can be linked with these, each reflecting the demand during a specific period;
- node without demand;
- fixed head node;
- reservoir.

Possible links are:
- ordinary pipe;
- check valve;
- pressure reducing and pressure sustaining valves;
- flow control valve;
- throttle control valve;
- pump;
- hydrant;
- gate valve.

The status of the links (open/closed) can be changed through an operation regime that is related to certain control times, pressures or flows.

Appendix E (section E.5) explains the principles of operation of the various system elements. The model building procedure and the layout of the accompanying files are described in Chapter 4.

The input files are created manually, with an editor. Modification of input files is done through the menu option “Edit / Edit Input” of ALEID2000. This is described in section 5.2.

2.5 Output presentation

Both input and output data can be displayed on the map of the network, by showing the ranges of the relevant parameter in different colours. A variety of output parameters can be presented in this way. The graphic snapshot results can be displayed for all calculated times.

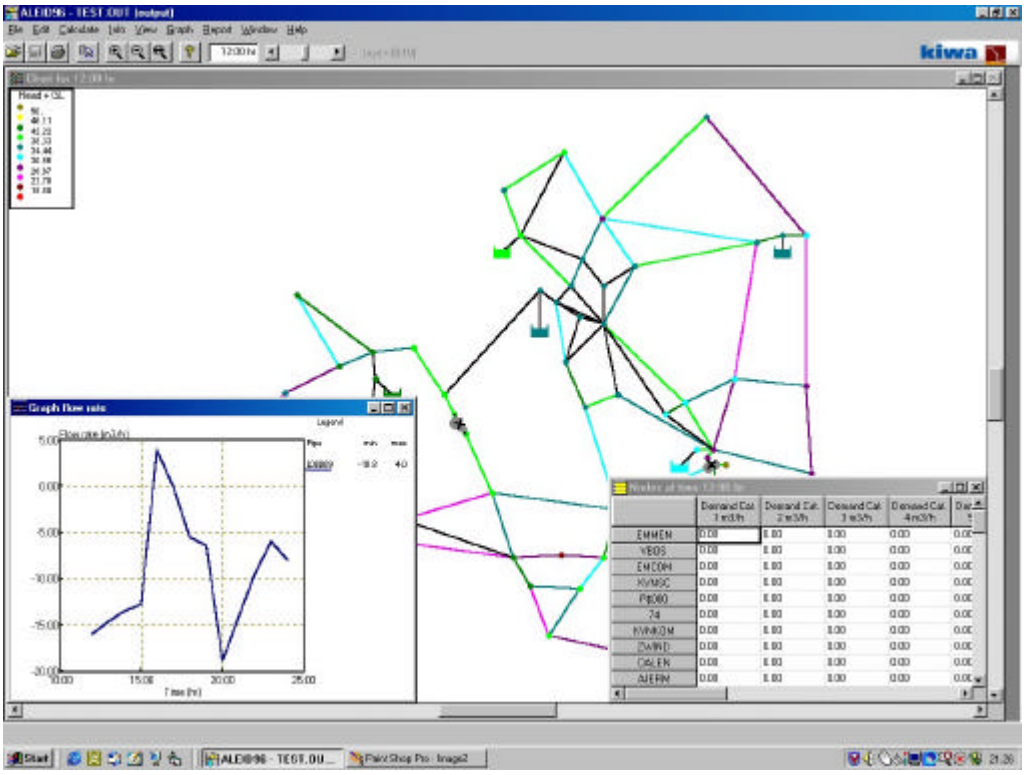


Figure 1 Presentation possibilities of ALEID2000

Moreover, diagrams can be created to present time series for certain parameters at a number of points in the network, e.g. variations in nodal pressure or pipe flow at certain locations, or the travel time towards certain points.

Finally, all results can be shown on screen in tabular form, and be saved as an ASCII file. All features are explained in Chapter 5, where the menu options are described in detail.

3 Installation and start-up of ALEID2000

3.1 System requirements

In order to run the program, the following minimum system requirements are necessary:

- an IBM compatible PC-486 or higher, supplied with:
  - a colour VGA or SVGA screen;
  - a minimum of 15 Mb of free space on the hard-disk for the ALEID2000 program itself, plus additional free space depending on the size of the network;
  - 16 Mb internal memory (RAM);
  - a floppy disk drive;
  - a mouse;
- Microsoft Windows 95 or better.

Recommended system requirements:

- Pentium (up to 1000 nodes), Pentium II (up to 2000 nodes), or Pentium III (over 2000 nodes);
- 1 Gb of free space on the hard-disk for the ALEID2000 calculation results
- 64 Mb RAM or more;
- a colour monitor of at least 17".

Network use (e.g. NOVELL) is possible, provided that DOS and Windows are supported.

3.2 Installation

ALEID2000® is provided on a single CD.

To install the program, do the following:

1. Insert the ALEID2000 CD in the CD-ROM drive; the CD will auto-run. Follow the instructions on screen.
2. If the CD does not run automatically, select the **Start** menu, then choose **Run**.
3. Type <drive>:\SETUP in the command box, where <drive> is the cdrom-drivename. Next click on the OK-button.
4. Default Aleid2000 will be installed in C:\ALEID2K. If you wish the ALEID2000 package to be installed in a different directory, give the full path name of your directory in the displayed dialog box.
5. Click the NEXT-button to proceed with the installation.

Upon completion, a new program group named ALEID2000 will appear on screen, with the ALEID2000 icon.

The files installed by this procedure are listed in Appendix C. If for any reason the package has to be re-installed, the above procedure must be followed again.

The programs in the package can be started by double clicking the ALEID2000 icon or by moving the cursor to the icon and pressing <ENTER>.

3.3 Adjusting settings after the installation

3.3.1 Language

The language setting in ALEID2000 is default English. The language can be changed at any time however (in the International version only!) in the **Edit**, **Options** menu.

3.3.2 Editor

The program ALEID2000 can make use of an ASCII editor. The desired default setting can be changed in the **Edit**, **Options** menu.

3.4 Working with ALEID2000

3.4.1 If you have no model

Before you can work with the program, a model has to be created first. See Chapter 4 for details. Rather than starting from scratch, it is recommended that you modify an existing model, such as the TEST model supplied on the ALEID2000 CD.

3.4.2 If you have an ALEID2000 model

If you already have an ALEID2000 model, ALEID2000 can be used directly. Open the ALEID2000 group in Windows, and double click the ALEID2000 icon. The program starts with an empty screen and the main menu. Before starting calculations, a project case must first be specified. This is done by clicking on **Open Project** command from the **File** menu. Next, the program reads the input data.

Now information can be requested (**Info** and **View** menu) or modified (**Edit** menu), and calculations can be performed (**Calculate** menu).

As soon as a calculation is done, the program imports the appropriate input into the shell. At that point the input becomes temporarily inaccessible. All output information can be requested (**Info**, **View**, **Graph** and **Report** menu).

The output is stored in a binary file (*.OUT). This file can be read only via the **Open Output** command from the **File** menu,. Once created, hydraulic calculations can be stored in a separate binary file *.HBI. Using this file, water quality calculations can be performed later. Here again, the input project has to be imported. For further information, see 5.6.

An extensive example of calculations is discussed in Chapter 6.

4 Building models in ALEID2000

4.1 Collection of relevant model information

As can be seen from the descriptions in the previous chapter, much data is required to build a model. To obtain this, various sources may be available in practice: maps, computerized databases (e.g. GIS-system), data obtained from monitoring system components (flows, pressures or energy consumption at the pumping stations, levels in the reservoirs), pressure and flow measurements at different points of the network (e.g. at the connection points between districts or neighbouring water supply companies), and finally the consumption data registered for billing purposes. The type of data needed depends on the type of the model that has to be created.

The first step is to set the objectives for which the model will be used. Possible purposes are:

- design or reconstruction of the infrastructure (transport model);
- network management (district model at the level of the main pipes);
- development of flushing programs (street level, 'one to one' model);
- study of network reliability (transport model);
- water quality considerations (transport or district model).

The purpose determines the level of detail required in the model.

Then the borders of the network have to be defined. For every system, a water balance should be formulated, so that all inflows and outflows of the model area can be inventoried and the local installations identified, such as pumping stations and storage facilities (see Figure 2).

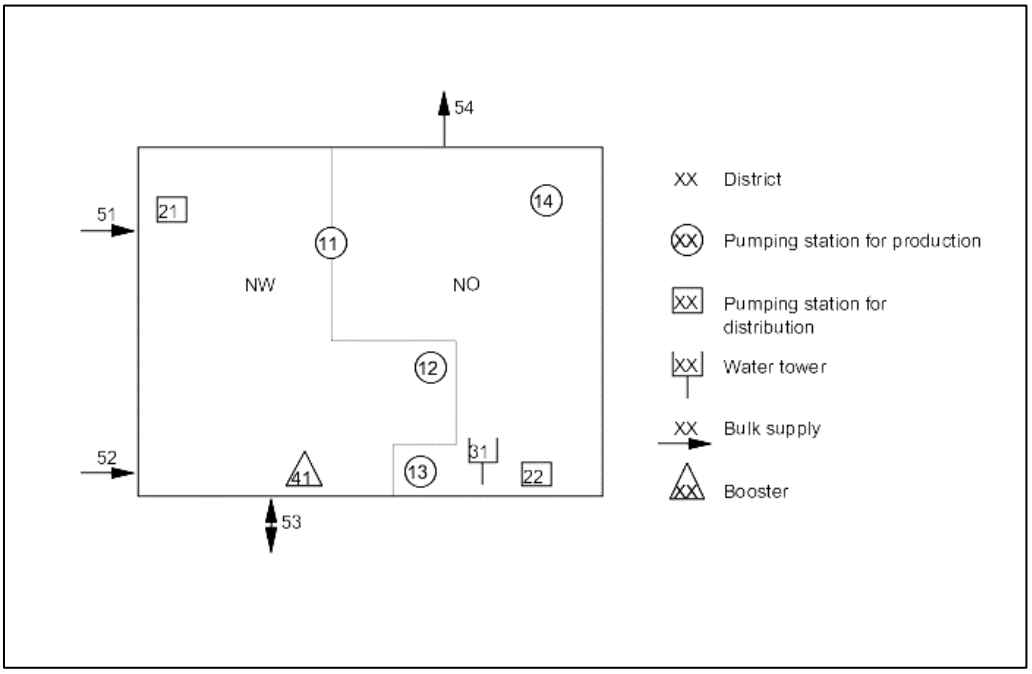


Figure 2 Schematised presentation of a water balance

Only then real data collection for specific model components can start.

4.2 Required input data

The input data required depends on the complexity of the model and the intended calculations. A minimum model configuration consists of one supply point (with known initial head), one or more demand nodes and one or more pipes. Depending on the available installations and the operation controls in the network, the model can be extended by:

- nodes without demand;
- pumps;
- reservoirs;
- hydrants;
- gate valves;
- various types of valves;
- demand variation patterns;
- additional settings;
- water quality information.

Appendix H presents a detailed overview of all data required and calculated for each system element.

4.3 Tools helping in setting up a model

Processing of the water company data required for model building sometimes can be made easier by making it compatible with the software used for modelling. Some examples from practice are:

Network information systems

Automated network information systems contain, in theory, all the network information relevant for the model layout. The problem here is that those systems usually provide a much higher level of detail than is necessary for modelling. Therefore, a criterion for selecting data from the information system has to be specified in one way or another. Further problems in data conversion may occur due to incompatible representation of the network appurtenances (e.g. valves) in the information system and in the modelling software.

The system layout can also be directly picked up from the available maps by using a digitizer.

AutoCAD

ALEID2000 creates network graphics based on coordinate inputs. Cad programs such as e.g. AutoCAD can be used for mapping the network information. However, it is not possible to import these drawings directly into ALEID2000.

Demand allocation

The demands registered for billing purposes have to be re-calculated in ALEID2000 as an average demand per node. If a model uses the standardised coordinate system for definition of node locations (in The Netherlands so called GBKN co-ordinates are in use), it is possible to make links between these co-ordinates and consumer's addresses available at the billing administration of the water supply company. To enable processing of such information, additional software to that for modelling should be used.

Automation of demand allocation is particularly important, bearing in mind annual variations; accurate demand distribution along the network model is crucial for its reliability.

4.4 Model validation and calibration

4.4.1 Calibration of the hydraulic model

Before a model can be applied in practice, it should prove to be sufficiently reliable. During the initial model building procedure precautions should be taken to prevent the following sources of errors in the input:

- underestimated effects of corrosion on the reduction of pipe diameters;
- other factors which cause a pressure drop, such as e.g. local losses due to bends or elbows;
- change in demand allocation (due to change of geographical borders, corrections of administrative errors, etc.);
- errors in peak factors, e.g. when a certain major user follows another demand pattern than the generally accepted pattern in the model, or when the effects of instantaneous demand are neglected in spite of the small number of consumers;
- operation control of the system components improperly translated into the model (e.g. pumping regimes or valve operation).

After the model input is determined with the best possible information available, the results of the first computer runs should be compared with different sets of measurements than those used in the model building process. Such validation provides an impression of the behaviour of the model.

If a model does not properly reflect reality (immediately or after some time), the model parameters should be modified based on network monitoring. This process is called calibration. Parameters that are often re-calibrated because their values are usually unreliable are:

- local (minor) loss factors;
- pipe roughness factors;
- demands;
- pump speeds.

Proper calibration of the hydraulic part of the model is very important. After all, the flows calculated by the hydraulic model are used as a basis for the water quality calculations. Hence, errors created by hydraulic calculations will affect the calculated water quality parameters as well.

4.4.2 Calibration of the water quality model

The calculations of the water quality model for conservative substances, fractions and travel times are solely dependent on the results of the hydraulic calculations. A well-calibrated hydraulic model will in principle lead to reliable water quality calculations. Problems can appear either at nodes where the assumption of complete mixing is questionable, or in the reservoirs.

The parameters used for non-conservative substances will need to be calibrated. The reaction model used in this case originates from American research about chlorine

distribution (see reference [2]). Nonetheless, further research is suggested into its application for these or other substances.

4.5 General remarks about the input files for ALEID2000

The layout of the input files is described in the following sections. The ALEID2000 input is split into a number of files, each with its own function. The types of files are:

- project file;
- model input files with:
  * basic information;
  * demand patterns;
  * co-ordinates;
  * control settings;
  * pump characteristics;
  * water quality parameters;
  * contour lines describing the topography;
- the initialisation file ALCALC.INI.

The following description will be given for each file type:
- identification;
- purpose;
- content;
- relations between the file elements;
- relations with other files.

Except for the project file, the general rule is that empty rows or rows starting with a semi-colon ";" will be skipped during data reading. If the semi-colon is placed somewhere within a row, the information following it will be treated as a comment used for clarification. Thus, it will not be considered as input for calculation. In any case, the total number of characters in a row should not exceed 200.

NOTE: Empty rows or those starting with ";" will automatically be removed after the input is modified and saved via the ALEID2000 menu.

In the project file, empty rows have a certain meaning and will not be removed from the file. For further explanation see the next section.

4.6 The project file

Purpose:
The project file consists of all the file names containing the model input information (i.e. the data required for calculation and presentation).

Identification:
Arbitrary file name, with extension .PRO

Description:
The project file has a fixed format of rows, as follows:
row 1 : project name / explanation
row 2 : name of the file with basic configuration
row 3 : name of the file with demand patterns
row 4 : name of the file with system control settings
row 5 : name of the file with water quality options
row 6 : empty row
row 7 : name of the file with co-ordinates
row 8 : name of the file with pump properties
row 9 : name of the contour file

- The project name / explanation can be up to 80 characters.
- A format description of the named files is given for:
  * basic file (*.BAS), in section 4.7;
  * demand patterns library (*.PAT), in section 4.8;
  * control settings file (*.STF), section 4.9;
  * water quality information file (*.WQI), section 4.10;
  * co-ordinates file (*.XY), section 4.11;
  * pump/hydrant/reservoir curves library (*.PHF), section 4.12;
  * contour file (*.CON), section 4.14.

- With the exception of .XY- and .PHF-files, it is allowed to have more than one file of each type in the same project (directory): more files of the same type distinguish possible alternatives in the network analysis.
- To be able to run a project, at least one basic and a co-ordinates file should be present. The remaining file types are optional, depending on the nature of the model. Thus, unlike other files, the project file may have some empty rows reserved for the file names. This will be interpreted by the program as 'File type not present'. Except for this purpose, no other empty rows should exist in the file.
- Rows or their parts starting with a semi-colon will be skipped during the file reading procedure.

Relations between file elements:
- The order of the file input described is compulsory.

Relations with other files:
- The project file contains the file names of the input files.
- The project file together with all the files it indicates should be placed in one (project) directory.

4.7 Model input file 1: Basic file

4.7.1 General

Purpose:
Description of the network layout and the main characteristics of its elements. These are:

- nodes (name, average demand, demand category);
- reservoirs (name, initial, minimum en maximum head, reservoir diameter or the reference to volume-depth relation);
- pipes (name, begin node, end node, diameter, length, roughness factor, local loss factor, presence of check valve);
- pumps (name, suction node, pressure node, reference to the file with pump/hydrant characteristics);
- hydrants (name, net node, outflow node, reference to the file with pump/hydrant characteristics);
- valves (name, begin node, end node, diameter, valve type, pre-set mode, local loss for fully opened position).

Identification:
Arbitrary file name, with extension .BAS

Description:
The basic file is divided into a number of sections, each starting with a fixed header, in square brackets. The file is concluded with an [END] row. An exact description of the content is given in the next section and in Chapter 7, Reference.

Relations between file elements:

- Uniform identification should be used for nodes and links. Being of the same nature, nodes and reservoirs make one list, just as pipes, pumps, hydrants and valves will form another group, all of these elements being considered as links between the nodes.
- Node names that appear in the list of links should already exist in the list of nodes.
- The order of the file input described is compulsory. See section 4.7.2.

Relations with other files:

- Node information: the basic node information is incomplete without additional data about node locations, i.e. X,Y co-ordinates and Z elevations, stored in the co-ordinates file.
- The type of the roughness factor depends on the head loss formula used. This must be defined in the Control settings file (*.STF), in case another than the default, the Darcy-Weissbach formula, is used.
- Identification numbers given for reservoirs, pumps and hydrants are references to the accompanying curves. These must be present in the curves library file (*.PHF) under the same number.
- Identification of the demand category given for each node refers to the demand patterns library file. The patterns there have to be declared under the same identification, except when the code 00 (= default setting) is used.

- The initial head given for each reservoir can be adjusted in the control settings file.

4.7.2 Set-up of basic configuration

The layout of the basic file is based on the principle that each component in it: node, pipe, valve, etc., should appear only once. For example, a pipe with a valve should be mentioned in the list of valves and not in the list of ordinary pipes; a reservoir node should not appear in the list of ordinary (discharge) nodes, etc. Modification of the file becomes easier in this way.

The basic file consists of the following data sections:

[TITLE]
[CONSUMPTION]
[TANKS]
[PIPES]
[PUMPS]
[HYDRANTS]
[VALVES]
[END]

NOTE: The order of the sections in the file is compulsory.

A minimum configuration for network calculation consists of nodes, pipes and fixed head nodes, i.e. the data in the sections [CONSUMPTION], [TANKS] and [PIPES], and the concluding section [END]. The file with co-ordinates should also be present (see section 4.11).

Detailed information on each section type is given in Chapter 7, in alphabetical order.

The program makes no distinction between uppercase and lowercase letters in the names of the model components.

4.8 Model input file 2: Demand patterns and factors library

4.8.1 General

Purpose:
Information about the demand categories i.e. (peak) factors that influence the average nodal demands specified in the basic file.

Identification:
Arbitrary file name, with extension .PAT

Description:
See the next section, as well as Chapter 7.

Relations between file elements:
- The factors multiply average nodal demands (see Appendix E, section E.3.3).
- The rows are identified by a pattern number. All data given in one row belongs to the same pattern.
- A time (calculation) step specified in the section [TIMES] is valid for all 24-hour patterns and for the concentration patterns from the water quality file.

Relations with other files:
- Relates to the basic file by adjusting average nodal demands with selected pattern(s).
- Name of demand pattern file is declared in the project file.

4.8.2 Set-up of demand pattern file

When available, the information given in the demand pattern file enables simulations of the basic model by applying various demand scenarios. A simulation can be defined as follows:

It is specified for a period beginning at 0 (hours), until a time T, determined by the number of factors. For 24-hour simulations, the standard time (calculation) step between two consecutive times (factors) is one hour. This can be modified separately.

During simulations longer than 24 hours, the multiplication of nodal average demands goes in two cycles: daily and weekly. Every 24 hours a particular peak factor will be multiplied with the related day factor. This is arranged in the program so that when all the factors of a certain pattern (row) are exhausted, the calculation proceeds with the first one again. The day factors are specified for a maximum of one week (7 values).

The average demand is further multiplied by the leakage, growth and correction factors. It is specified in the basic file which patterns are valid for certain nodes.

Different demand patterns can be attached to one model area, e.g. for various demand categories and/or minimum, maximum or average supply conditions.

The demand pattern file consists of the following data sections:

[TIMES]
[PATTERNS]
[DAYFACTORS]
[LEAKAGE]
[CORRECTION]
[RELATIVE GROWTH]

NOTE: The order of the sections in the file is compulsory.

Detailed information on each section type is given in Chapter 7, in alphabetical order.

No minimum configuration for network calculation is required for this file. Regarding the comments in the file the same applies as in the case of the basic file.

4.9 Model input file 3: Status and control settings file

Purpose:
This file contains all information about control modes that influence the model or program execution. These are:
- pressure, time or flow dependent control of components from the basic file;
- hydraulic options, such as type of head loss formula;
- nature of pressure related demand.

Identification:
Arbitrary file name, with extension .STF

Description:
The file content is divided into the following sections:

[CONTROLS]	control settings;
[PRESS.DEPENDENCE]	pressure dependence of nodal demand;
[OPTIONS]	various options.

Detailed information on each section type is given in Chapter 7, in alphabetical order.

Relations between file elements:
- The given controls affect the model, as specified. The user should know and take care that the chosen control settings simulate reality properly. The program does not optimise any of the control settings.
- The data for individual nodes given under the [PRESS. DEPENDENCE] section takes priority over the GLOBAL specified value.

Relations with other files:
- Related to the basic file by attaching the control and pressure dependent demands as selected there.
- Name of the control settings file is declared in the project file.

4.10 Model input file 4: Water quality options

Purpose:
Information about water quality options in the model.

Identification:
Arbitrary file name, with extension .WQI

Description:
The file consists of the following sections:

[QOPTIONS]	sets the water quality calculation type
[QUALITY]	gives initial concentration or (source) % in a node
[REACTIONS]	gives reactions per pipe
[SOURCES]	indicates sources of substances

[QPATTERNS] specifies variation patterns of source concentration

Relations between file elements:
The calculation type chosen under [QOPTIONS] determines which sections will be required further in the file. The [QUALITY] section can be present in all options, whereas the other headers will be used for concentration calculations only. The pattern numbers specified under [SOURCES] have to be declared under the [QPATTERNS] header.

Detailed information on each section type is given in Chapter 7, in alphabetical order.

Relations with other files:
- Relates to the basic file by referring to the node names existing there.
- Name of the water quality file is declared in the project file.
- The time (calculation) step for [QPATTERNS] is the same as used for the demand patterns from the demand pattern library.

4.11 Model input file 5: Co-ordinates file

Purpose:
Information about node locations in the model area. X and Y co-ordinates together with node elevations (Z) are specified.

Identification:
Arbitrary file name, with extension .XY

Description:
The co-ordinates file has only one section: [COORDINATES] . For details see section 7.3.

Relations between file elements:
- The node names must be unique.

Relations with other files:
- It is allowed to keep only one co-ordinates file per directory (project). All basic files (if more than one) can make use of this file.

4.12 Model input file 6: Curves library file

Purpose:
To describe main characteristics of pumps, hydrants and reservoirs in the model area.

Identification:
Arbitrary file name, with extension .PHF

Description:

The file consists of three sections:
[PUMP CURVES],
[HYDRANT CURVES] and
[TDATA].

For a detailed description see Chapter 7.

Relations between file elements:
- The identification numbers in one section must be unique.

Relations with other files:
- It is allowed to keep only one curves library file per directory (project). All basic files (if more than one) can make use of this file.

4.13 Initialisation file ALCALC.INI

Purpose:
Information that does not directly influence the particular model, such as the names of the output files, definition of the times and time steps. The file ALCALC.INI will be automatically created once calculation by ALEID2000 is being requested. The file input is created from the values specified in the "Run - Standard" menu.

Identification:
Fixed name, ALCALC.INI

Description:
The file consists of the following sections:

[GENERAL]	data about the simulation run parameters
[SOLUTION]	names of the output files for hydraulic calculation
[REPORT]	output data used for presentation of the results by the Windows shell.
[QTIMES]	data required for generating a water quality solution (optional).

For a detailed description see Chapter 7.

Relations between file elements:
- No special requirements exist

Relations with other files:
- ALCALC.INI contains the names of the binary output files.

4.14 Model input file 7: Contour file

Purpose:
Description of the topography in the model area. The contour can consist of several lines. Each row contains one set of X,Y co-ordinates or the asterisk character. The

contour will be drawn starting with co-ordinates from the top to the bottom of the file. After reading the asterisk ("*") character, the program proceeds by drawing a new line from the following co-ordinates.

Sections:
There is one section with the name [LINE].

Format:
X Y
or
*

Parameters:
- X and Y are co-ordinates, which have to be specified in the same co-ordinates system as those used for the nodes.
- The X,Y-rows can be split by the partition character "*" (asterisk).

Remarks:
none

4.15 Measurements data file (menu option “Graph - Options”)

Purpose:
In the menu option “Graph - Options” a set of measurements can be specified for the related graph. This data is read from this file.

Format:
Xi Yi

Parameters:
Xi : value for point "i" of the parameter on the X-axis of the graph
Yi : value for point "i" of the parameter on the Y-axis of the graph

Remarks:
- Each row in the file may contain one pair of points only.
- In theory, the extension of the file is free. To differentiate it from the extensions of other files, the extension *.DAT is recommended. The program uses *.DAT as the default setting of **Graph**, **Options**.

5 Working with the program

5.1 Menu conventions

After the program is started, a screen with the main ALEID2000 menu appears. This menu remains active during the entire program. Each option from the main menu branches into a pull-down menu with program commands. Selected options (commands) from these menus will either be instantly executed or a dialog box with further instructions will follow.

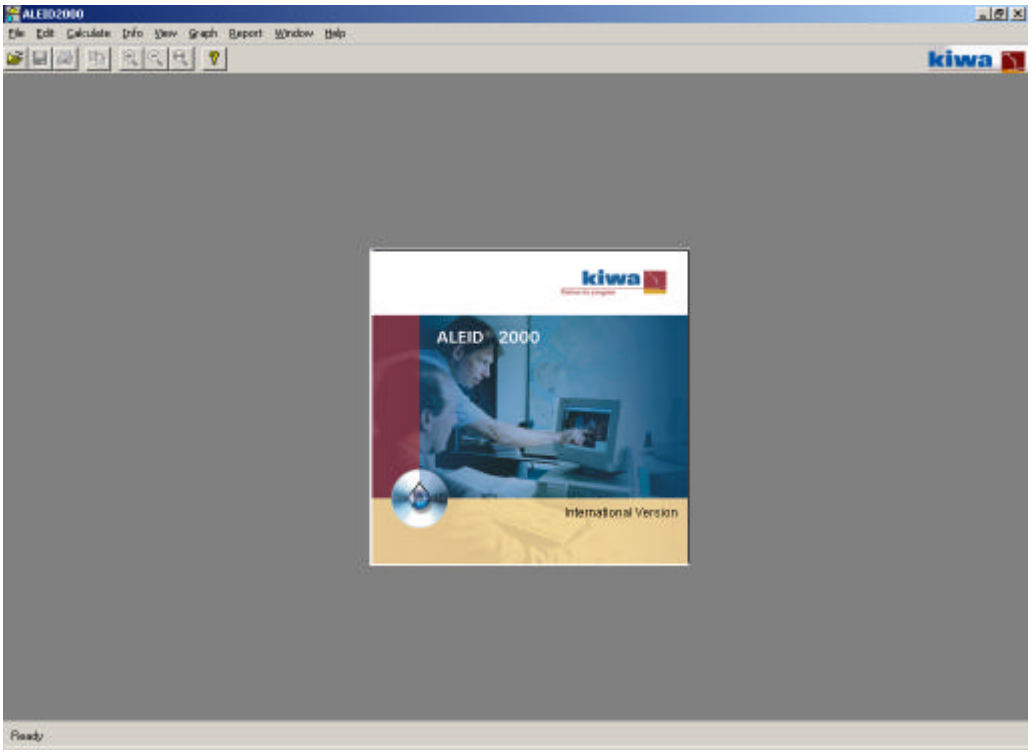


Figure 3 Opening screen of ALEID2000 UK version

Options in the menus can be selected with the mouse or from the keyboard. Clicking the mouse on a certain option will activate it. If the keyboard is used, the pull-down menus become active if the combination <ALT-*> is pressed simultaneously, where "*" is one of the underlined letters from the main menu. An option from the pull-down menu will then be executed by typing the corresponding underlined letter. If the cursor pad is used, a highlighted option will be activated by pressing <Enter>.

Menu options displayed in grey ('greyed out') are temporarily disabled. This means that the information required for these options is at that moment inaccessible to the program. Sometimes menu options are accessible (visible) only if certain conditions are satisfied. For example, the input of water quality parameters is only possible if a water quality file was already retrieved.

Pressing the “OK” button in a dialog box means that the program proceeds with the information specified in that box. Choosing the “Cancel” button will quit the dialog box without saving the latest modification.

5.2 Overview of the menu options

An overview of the menu options in ALEID2000 is given in the tables below. In Table 1 the first command in each column comes from the main ALEID2000 menu, being followed by the available options in the corresponding pull-down menu. (see also Figure 3).

Table 1 Main menu ALEID2000

File	Edit	Calculate	Info	View	Graph	Report	Window	Help
Open Pro-ject	Editor	Standard	All Nodes & Pipes	Zoom In	Time Series Node	Any File	Tile Windows Horizontally	About ALEID 96
Open Output	Edit Input		Nodes	Zoom Out	Time Series Pipe	Summary Output	Tile Windows Vertically	
Save Input	Copy		Pipes	Zoom All	Demand	ASCII Output	Cascade Windows	
Project Summary			Summarize Demand	Clear Map	Curve	Snapshot	Arrange Icons	
Printing			Contour	Colour Coding	Options	Time Series		
Print Setup			Save Macro	Show Legend				
Exit			Read Macro	Change Legend				
				Toolbars				
				Options				

A number of menu options branch out later on. These are:
- **Edit Input** (see Table 2)
- **Contour**, with sub-options **On** and **Off**
- **Show Legend** and **Modify Legend**, both with sub-options **Node** and **Pipe**
- **Toolbars**, with sub-options **Standard** and **Toolbox**
- **Snapshot** and **Time Series**, both with sub-options **Node** and **Pipe**.

Table 2 Edit Input: Sub-menu options

Edit Input ?	Node ?	Edit Add Remove Move
	Link ?	Edit Add Remove
	Demand Patterns	
	Controls	
	Curves	
	Hydraulic Options	
	Water Quality Options	
	Concentration Patterns	
	Contour	

Not all of the menu options are accessible at any specific moment. This depends on the type of retrieved information and the window in use.

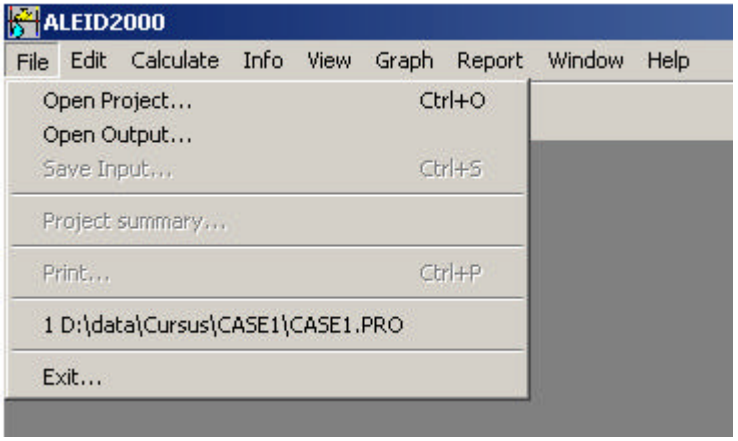


Figure 4 The ALEID2000 main menu

5.3 Window types

More windows can be activated at the same time by using the ALEID2000 menus. The window with the network layout is usually displayed, along with the main menu. Moreover the following windows can be opened:

- graph window;
- table window;
- viewer window;
- DOS window for editing or calculation purposes.

Some menu options are accessible only through these special windows. Those will be listed later.

5.4 File menu

5.4.1 Open project

When this option is selected, the program closes all current screens and retrieved information, and asks to open another project. The standard 'Open Project File' window with the list of available sub-directories will be opened in this case (see Figure 5). All projects (see section 4.6) listed in files with the .PRO extension can be selected from the menu.

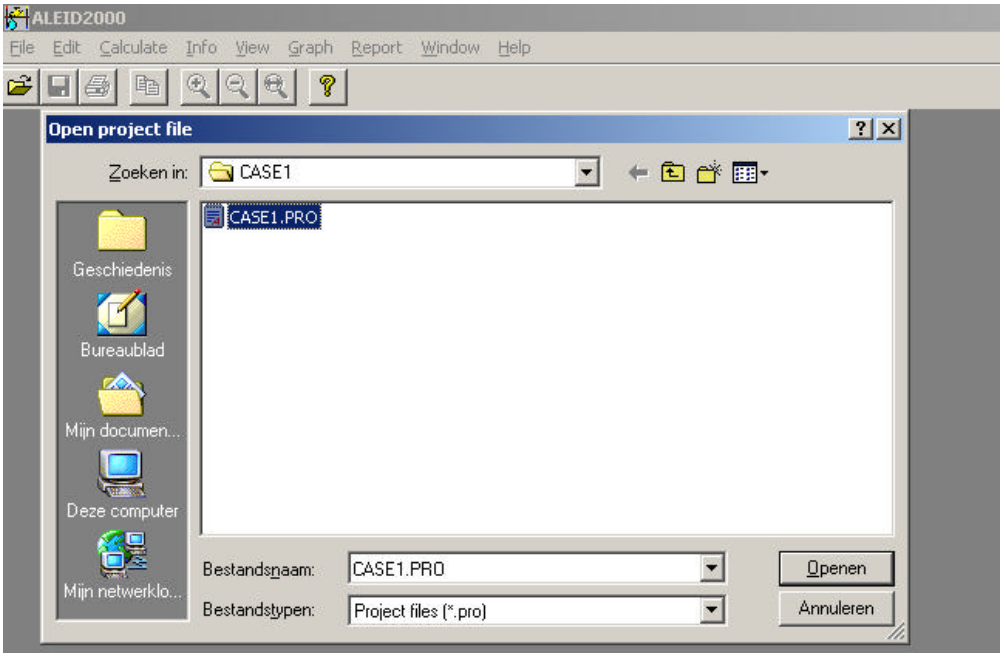


Figure 5 Opening a project file through the standard 'Open Project File' window

One project consists of the following files:

- a basic configuration file;
- a control settings file *[optional]*;
- a demand patterns library *[optional]*;
- a water quality file *[optional]*;
- a pump / reservoir library *[optional]*;
- a co-ordinates file;
- a contour file *[optional]*.

A minimum requirement for program execution is a project consisting of a basic and co-ordinates file.

After a project is chosen, the content of the selected .PRO file, i.e. the list of available input files in the project, will be shown on the screen (see Figure 6). It is possible here to modify an existing project by adding or removing certain files from the list. This is important in particular for files that have not yet been created. By specifying a non-existing file, the program will open an empty file with the given name, which allows access to the **Edit** menu and the **Edit Input** submenu. If you wish to modify one of more filenames, select the **Save, then open** option after clicking the arrow next to the **Open** button. By default, the **Open** option is activated.

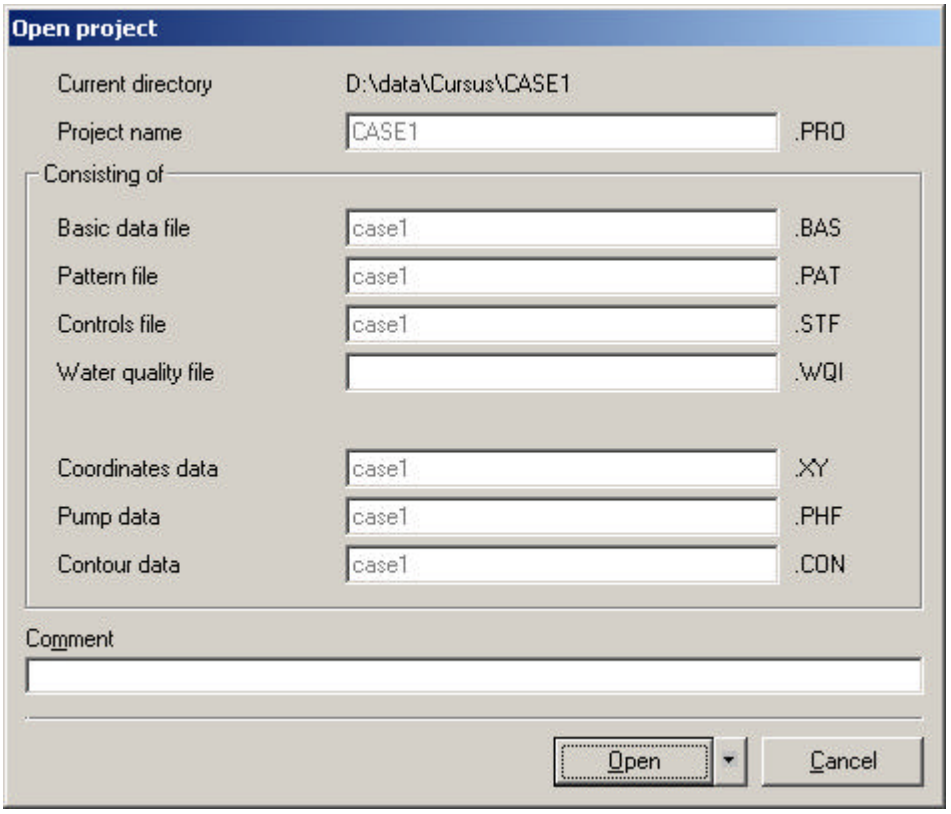


Figure 6 The 'Open Project' dialog box

Thus, there are two possibilities:

- The project will be retrieved without modification. The retrieved information will be presented in the network window.
- The project file will be modified, saved (possibly under another name) and retrieved. The retrieved information will be presented in the network window.

After being retrieved, the network layout will be presented with pipe diameter and nodal ground elevation ranges shown in different colours (see figure 7). The work on the model may start now by choosing some of the options from the main menu required for presentation or editing of the input data.

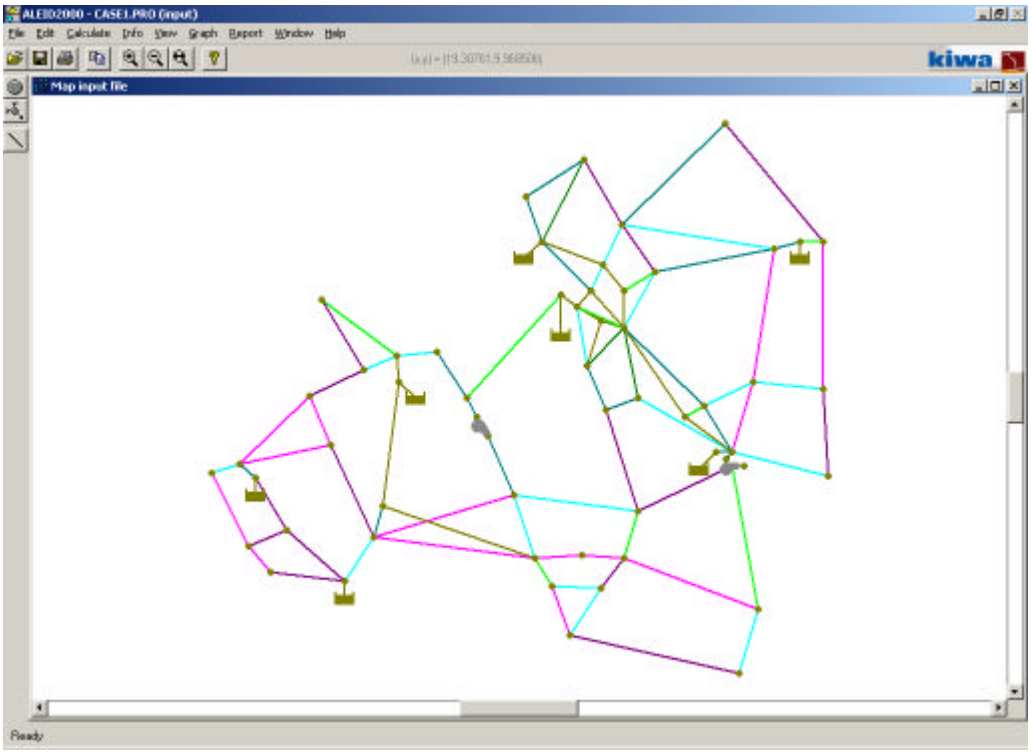


Figure 7 Presentation of the network lay-out (input)

The **Open Project** option is accessible at any moment. Its implementation has no effect on other files except the project file(s), *.PRO.

5.4.2 Open Output

This option makes the results of calculation available for presentation on screen. A file with the extension .OUT selected from the file window will be opened in this case. This is a binary type file, thus not directly readable (as a table). To store the results in files with readable format, choose **ASCII Output** from the **Report** menu.

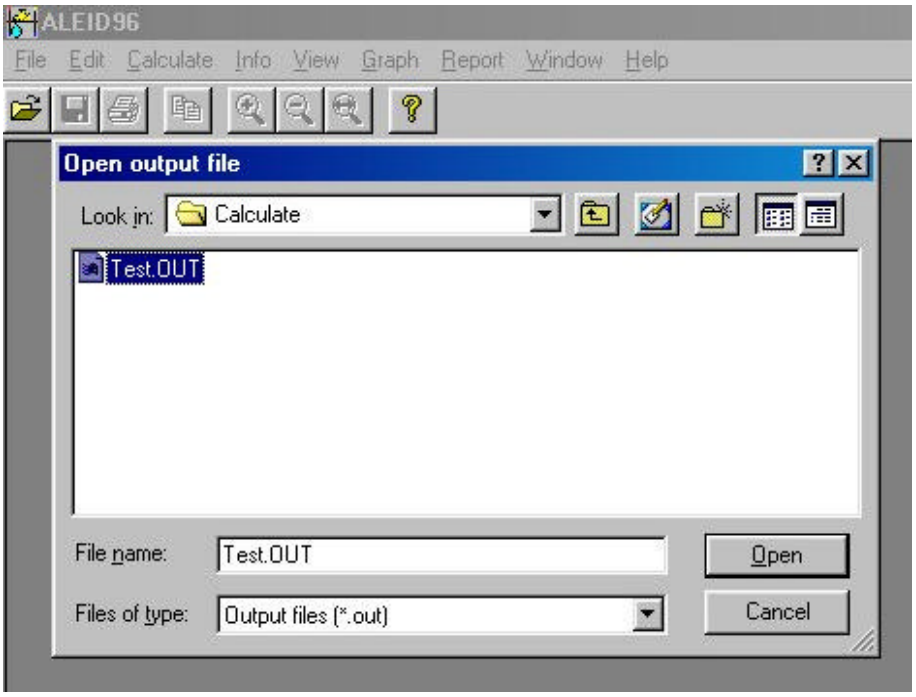


Figure 8 The 'Open Output File' dialog box

The information from the output file will be shown on the map of the network. This is usually a pressure situation in the network being presented in different colours for different ranges, or any other parameter chosen (and automatically saved) during the previous screen presentation. The method of graphic presentation is similar to that for input information, however with more (menu) options being available in this case. These options become accessible only after a certain .OUT file is retrieved.

The content of a contour file will also be retrieved if such a file exists in the project.

The **Open Output** menu is accessible at any moment.

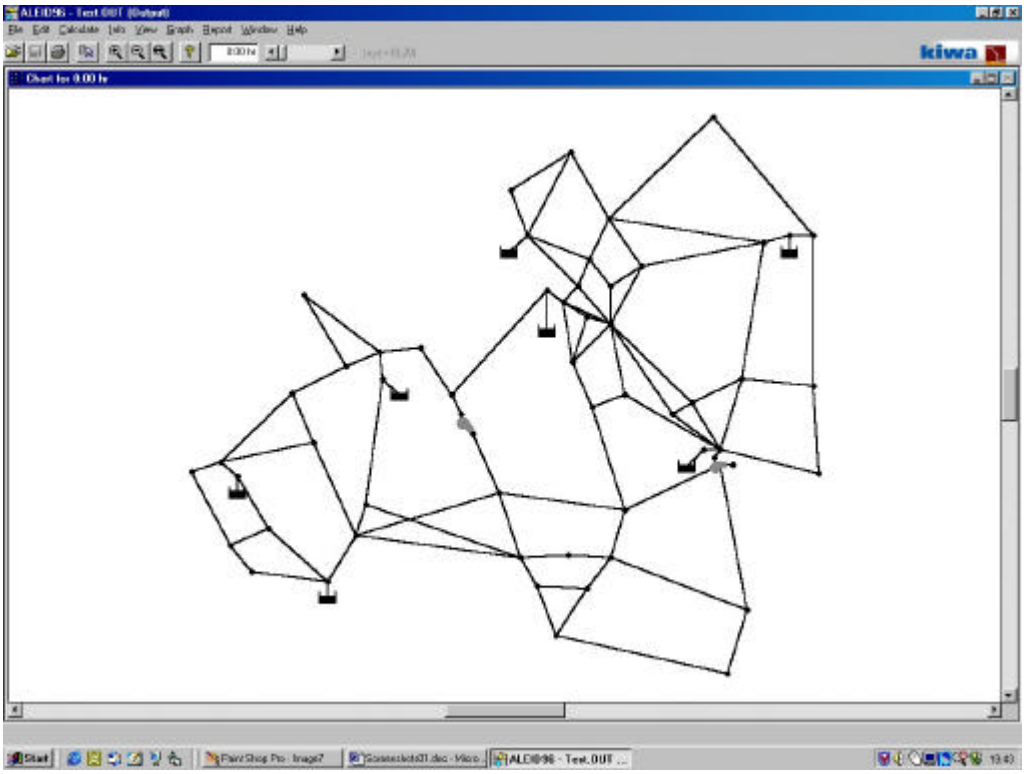


Figure 9 Typical screen after opening output file

5.4.3 Save Input

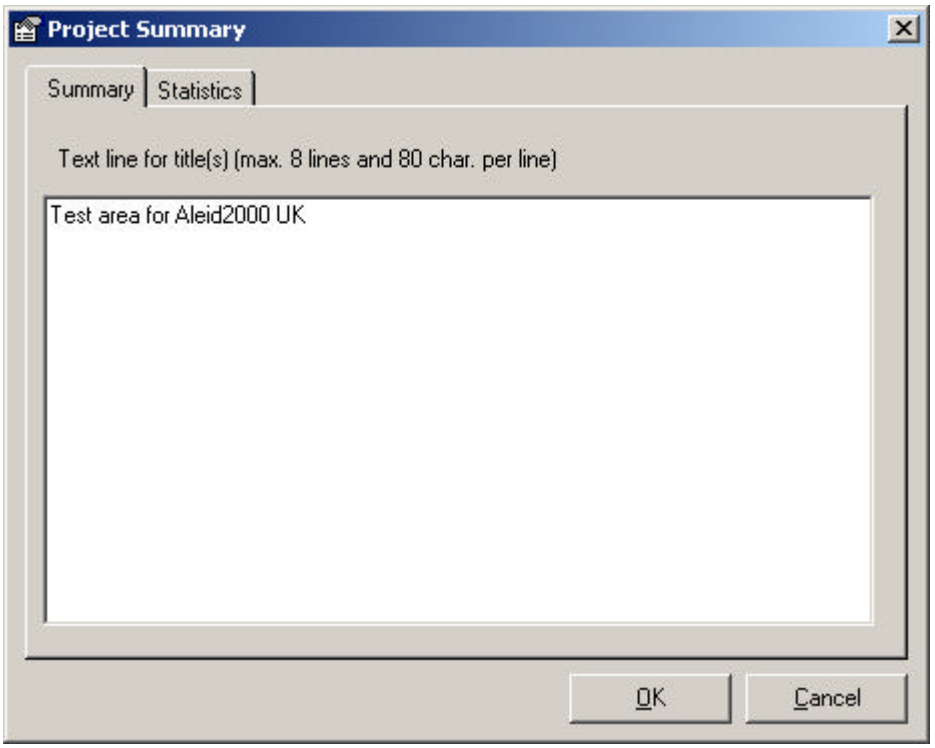
This command saves all modifications in the input files, done via the **Edit Input** submenu of the **Edit** menu. Here it is also possible to establish a new project by changing the project name or the file names in the displayed list.

NOTE: Only modifications done through **Edit Input** (**Edit** menu) will be saved by the **Save Input** option. Additional comments, empty rows or specific layouts in the input files will be ignored.

5.4.4 Project Summary

This command opens a window with two sub-windows: **Summary** and **Statistics**. The first sub-window, **Summary**, allows comment lines to be added to the input file. Up to 8 lines, each with a maximum of 80 characters, are allowed. Additional lines or characters will be ignored and thus will not be saved.

The second sub-window, **Statistics**, indicates the numbers of nodes, reservoirs, pipes, pumps and valves in the network as defined by the input file.



'Figure 10 Summary' sub-window of Project Summary option

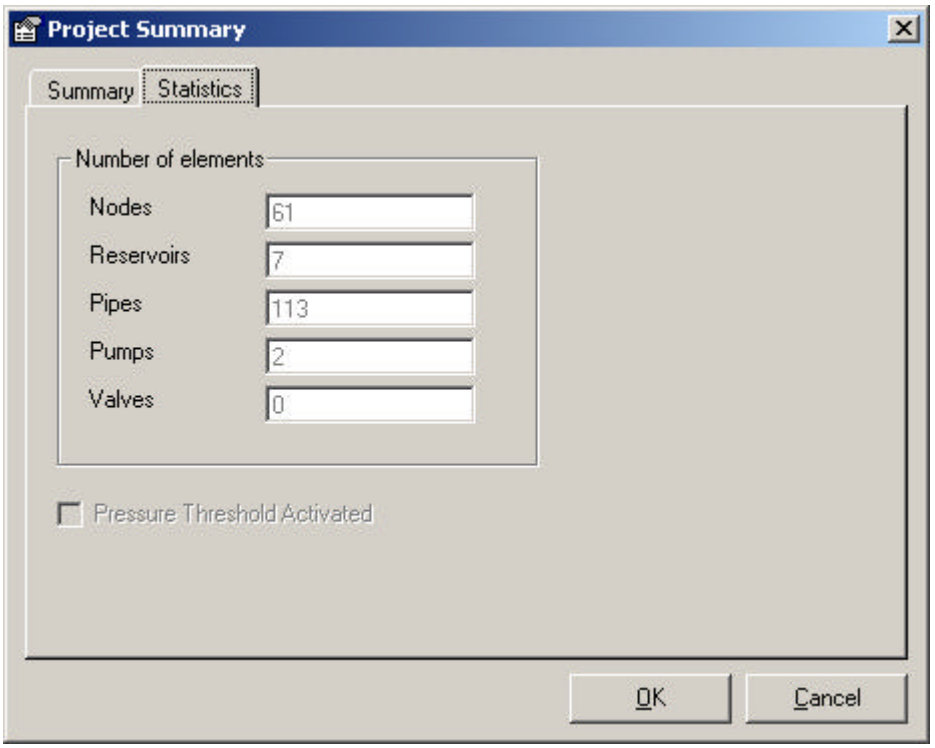


Figure 11 'Statistics' sub-window of Project Summary option

5.4.5 Printing

This command makes a print of the active window by the selected printer. All windows opened by using options **View**, **Graph** and **Report** (including the viewer) from the main ALEID2000 menu can be printed.

Before the printing actually starts, the printer properties, print range and number of copies can be set. The printout orientation is selected from the **Printer Settings** menu option (see later). Network printouts in vertical ('portrait') orientation allow for a few more lines of text.

5.4.6 Print Setup

The settings defined by the **Print Setup** menu determine the printout format in general. By choosing this option a standard window will be activated, allowing for the definition of:

- printer type/name
- printer properties
- paper size
- paper source
- print-out orientation: portrait or landscape

5.4.7 Exit

This command terminates the program, keeping the current settings saved in the main initialisation file, ALEID2000.INI. These settings are:

- parameter shown in colours;
- node size / link thickness;
- legend (colour) settings;
- background colour;
- form of highlighting;
- track of the data files in use before exit;
- aspect ratio (width/height proportion in the graphs).

5.5 Main menu: option "Edit"

5.5.1 Editor

This command starts a DOS-editor loading a file chosen via the standard file window. The specified editor operates completely independent from the program.

Any DOS-editor can be used if declared in the file EDITOR.PIF. For further details regarding the editor installation refer to section 3.3.2.
The **Editor** option is accessible at any moment.

5.5.2 Edit Input

A model input created via the program menus and/or by network building directly on the graph, can be modified by this option. One must know the origin of the information that has to be modified, i.e. from which project file the input is taken. An input file, e.g. the control settings file, which was not specified in the retrieved project cannot be modified.

This option can be accessed only from the map-window, after an input project file has been retrieved. For further details about the input format, refer to Chapter 4.

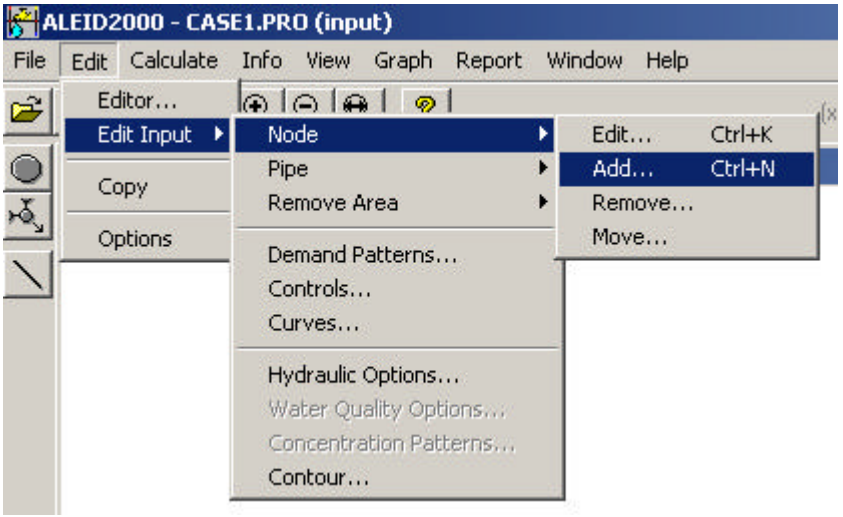


Figure 12 The 'Edit Input' menu

By clicking on the **Edit Input** option a pull-down menu with available options will appear. Some of these options branch further (see Figure 12).

5.5.2.1 Edit Input - Node

In this menu option, the network nodes can be re-arranged directly on the map. Available sub-options are **Edit**, **Add**, **Remove** and **Move**. After choosing any of these, the mouse cursor turns into selection mode i.e. transforms into a vertical arrow ready for selection of a particular node or a new location. The same result can be obtained by right-clicking on an element in the network without first selecting the **Edit Input** menu.

Edit

Clicking on a network node (in the cursor selection mode) opens a dialog box with current information about the node. This is, depending on the node type:

Node

- name
- number of existing categories
- average demands for each category
- code of hourly, daily, leakage, correction, and growth factors pattern attached to each average demand
- comment
- ground elevation
- X and Y co-ordinates
- initial concentration + comment
- source concentration + comment + pattern number

Reservoir

- name
- initial head
- type with additional parameters where required
- comment
- ground elevation
- X and Y co-ordinates
- initial concentration + comment
- source concentration + comment + pattern number
- reservoir reaction - coefficient + comment

All indicated information can be modified.

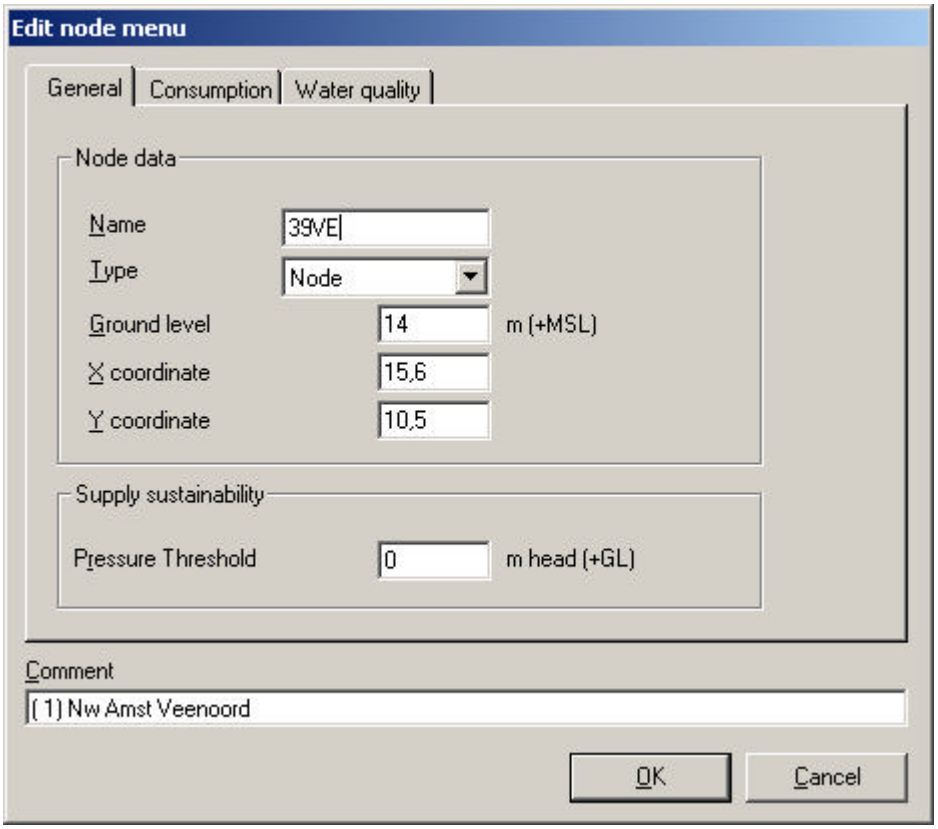


Figure 14 Edit node menu; 'General' tab

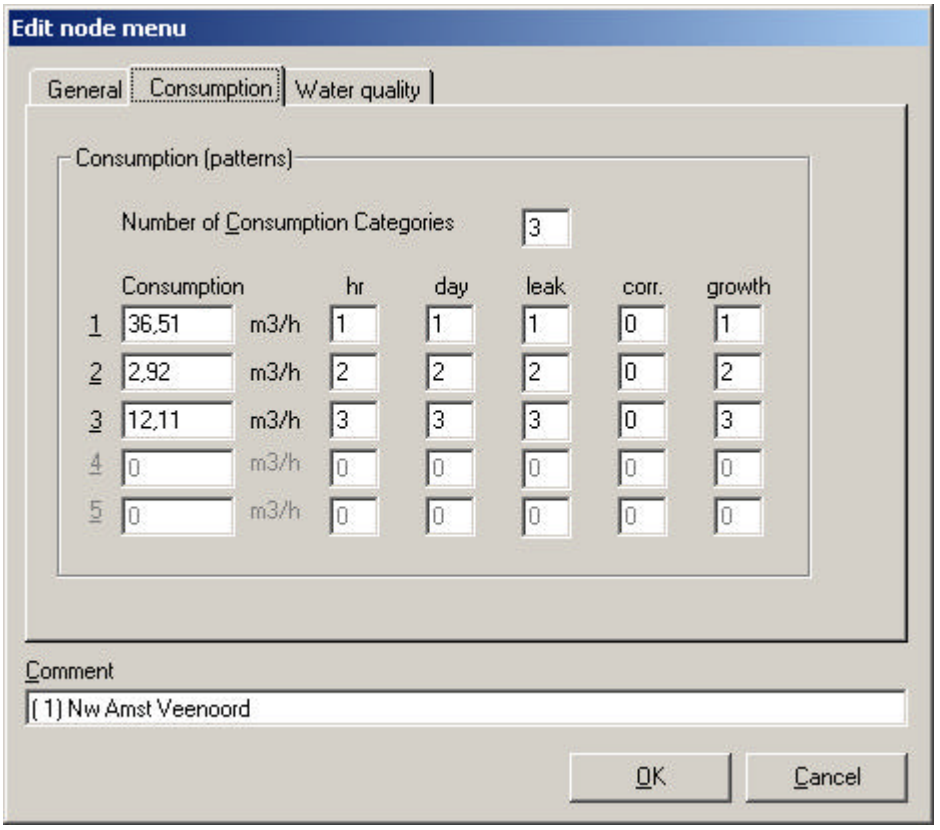


Figure 15 Edit node menu; 'Consumption' tab

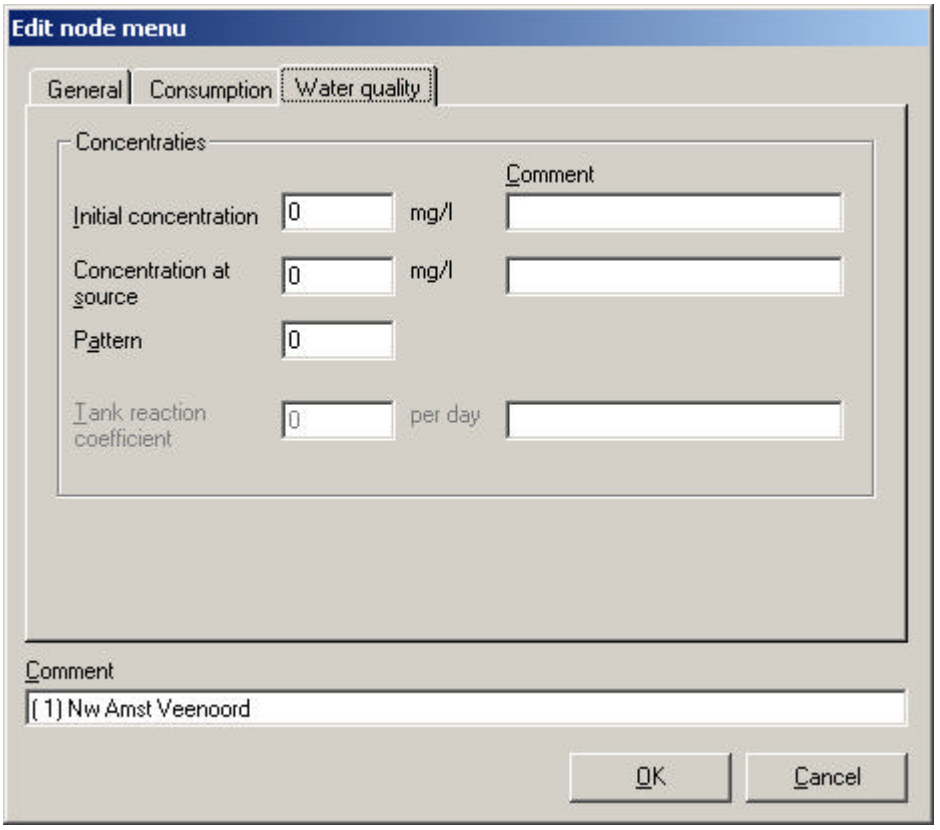


Figure 16 Edit node menu; 'Water quality' tab

The following limitations exist:

- a node name can have a maximum of 7 characters;
- the new name should not exist amongst the other nodes/reservoirs;
- the number of demand categories attached to the node should range between 0 and 5;
- the pattern identification numbers should range between 0 and 99.

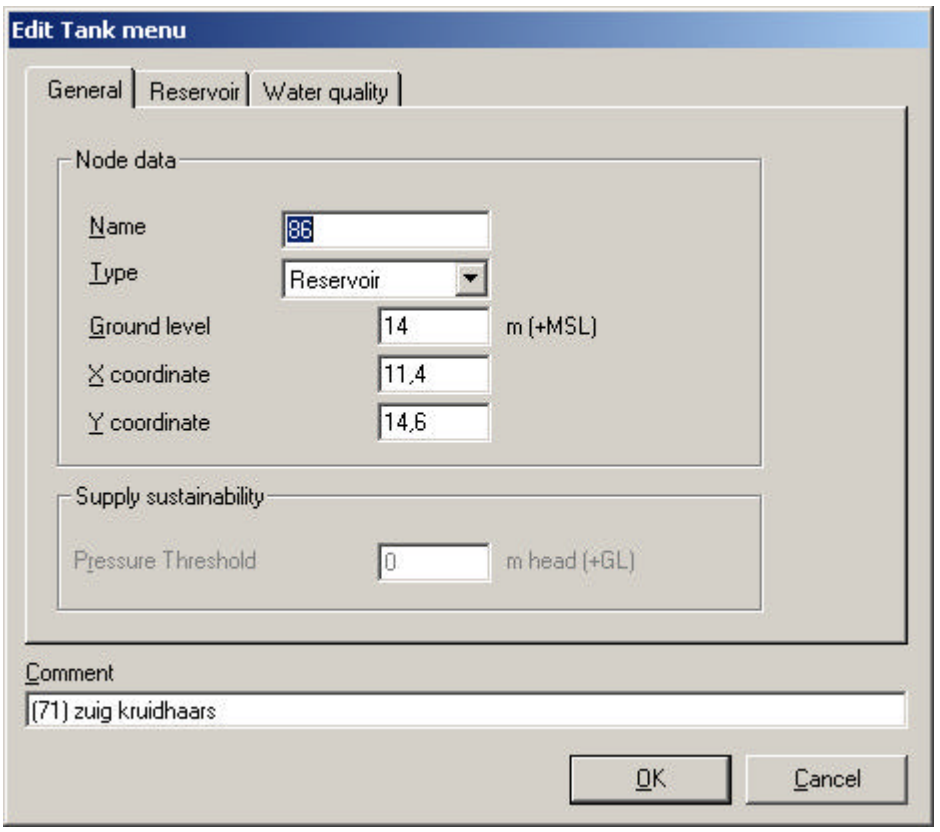


Figure 17 Edit Tank menu, 'General' tab

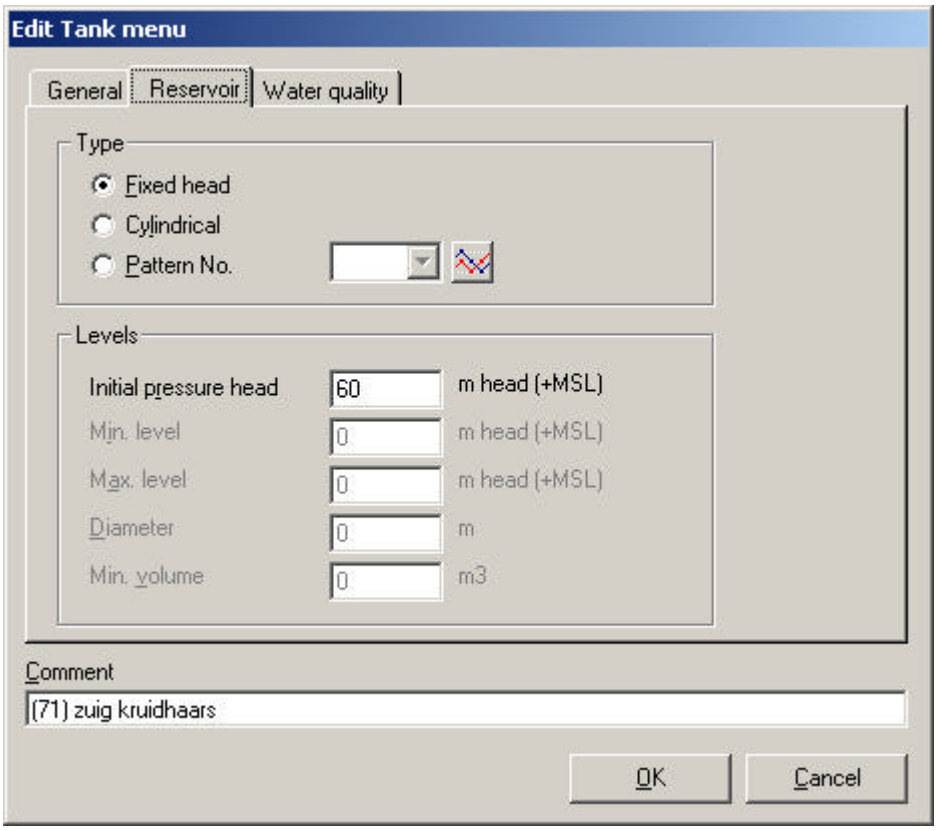


Figure 18 Edit Tank menu, 'Reservoir' tab

Add

After running this option, the location of the new node has to be determined first. By clicking in the selection mode at a specified position on the map, the same dialog box appears as in the case of the **Edit** option (see Figures 14-16). The required information should be filled in.

A short-cut for adding a node is provided by one of the buttons immediately to the left of the top of the screen:

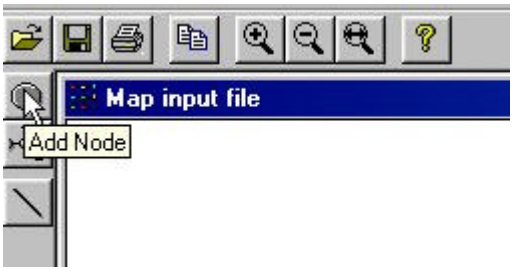


Figure 19 'Add node' button

Remove

A selected node will be deleted from the system by selecting this option. The action has to be confirmed by pressing the "OK" button.

NOTE: All links connected to the removed node will be removed from the network as well.

Move

With this command a new location of the existing node can be specified by clicking in the selection mode. A window appears on screen, requesting confirmation of the **Move** command.

A short-cut for adding a node is again provided by one of the buttons immediately to the left of the top of the screen:

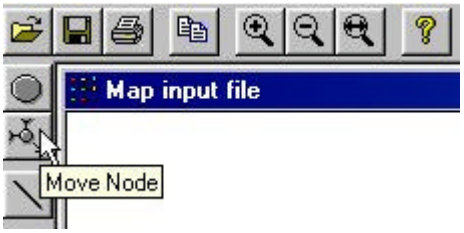


Figure 20 'Move node' button

Related files
The network modification done by the **Edit Input**, **Node** option is temporary: it is registered in computer memory only as long as the program is active. If it is preferred to make it permanent i.e. registered also in the input files, choose **Save Input** from

the **File** menu (see section 5.4.3). This action will affect the content of the following files:

- Basic file - sections [CONSUMPTION] and [TANKS]
- Co-ordinates file
- Water quality file - sections [QUALITY], [REACTIONS]-TANK and [SOURCES]

5.5.2.2 Edit Input - Pipe

In this menu option, the network links can be re-arranged directly on the map. Available sub-options are **Edit**, **Add**, and **Remove**. After choosing any of these, the mouse cursor turns into selection mode i.e. transforms into a vertical arrow ready for selection of a particular pipe, or for node locations for a new pipe. A similar effect can be obtained by right-clicking directly on the selected element of the network. A small selection window then appears on screen:

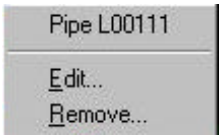


Figure 21 Selection screen after right-clicking directly on a network element

Edit

Clicking on a network link in the selection mode returns the question: 'Edit link <link name> ?', where <link name> is the name of the chosen link. By pressing "OK" a dialog box with current information about the link will appear. This is, depending on the link type:

Pipe / Check valve
- name
- name begin node
- name end node
- diameter
- length
- wall roughness
- local loss factor
- comment
- bulk reaction coefficient + comment
- wall reaction coefficient + comment

Valve (PRV, PSV, PBV, TCV, FCV)
- name
- name begin node
- name end node
- diameter
- valve characteristics
- local loss in open position
- comment
- bulk reaction coefficient + comment
- wall reaction coefficient + comment

Pump / Hydrant
- name
- name begin node
- name end node
- number of attached pump/hydrant curve

- comment
- bulk reaction coefficient
  + comment
- wall reaction coefficient
  + comment

All indicated information can be modified.

The following limitations apply:
- a link name can not have more than 7 characters;
- the new name should not exist amongst the other links;
- the diameter has to be greater than 0 mm;
- length, wall roughness and local loss factor should be equal to or greater than 0;
- the pattern identification numbers should be between 0 and 99;
- no other link types than those named in the list are allowed;
- the specified numbers of pump/hydrant curves must exist in the pump/hydrant library (.PHF file).

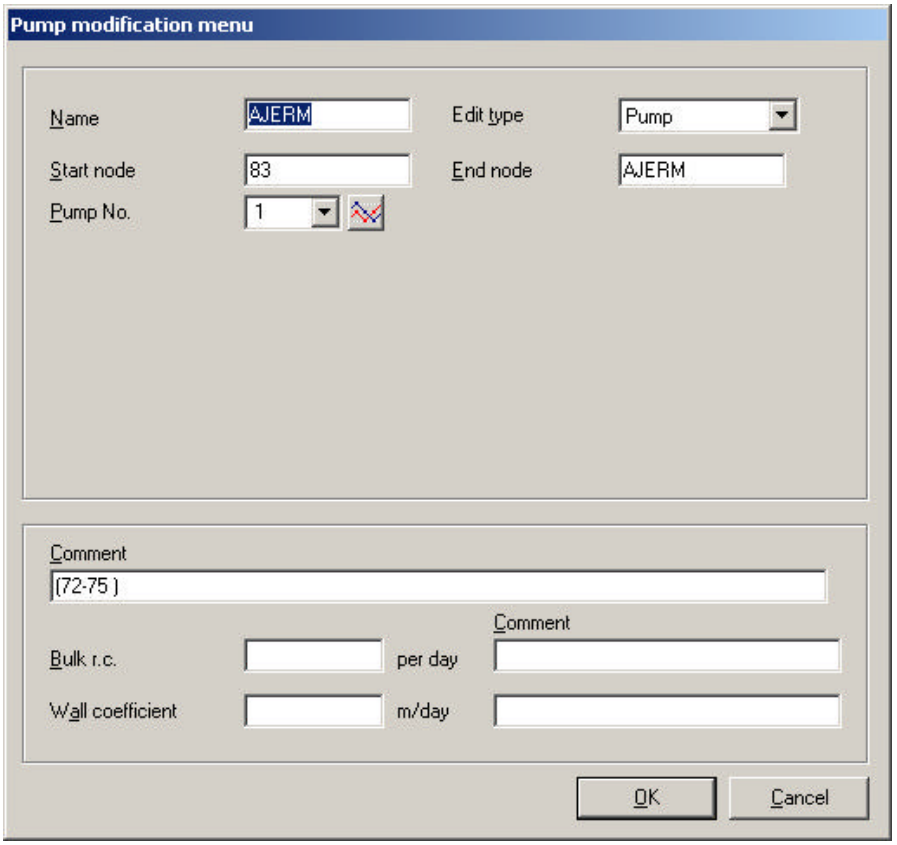


Figure 22 Pump modification menu

Add

After activating this option, the location of the new link has to be determined first. By clicking, in the selection mode, at positions chosen for the link nodes, the same dialog box appears as in the case of the **Edit** option. The required information should be filled in.

Remove

The selected link is deleted from the network by activating this option. The action has to be confirmed by pressing the “OK” button.

NOTE: The link node without other connections to the network will be removed together with the link.

Related files
The modification done by the **Edit Input**, **Pipe** option is temporary: it is registered in computer memory only as long as the program is active. If it is preferred to make it permanent i.e. registered also in the input files, the menu option **File**, **Save Input** should be chosen (see 5.4.3). This action will affect the content of the following files:

- Basic file - sections [PIPES], [PUMPS], [HYDRANTS] and [VALVES]
- Water quality file - sections [REACTIONS]-WALL and BULK

5.5.2.3 Edit Input – Demand Patterns

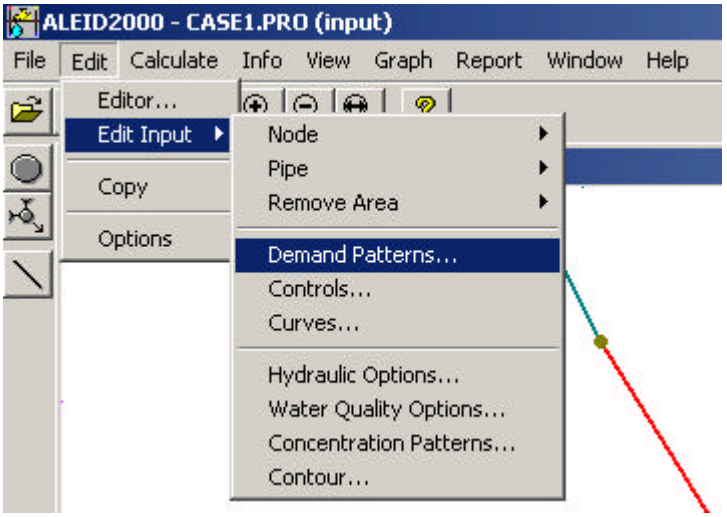


Figure 23 Selection menu for option ‘Edit Input – Demand Patterns’

When this command is selected, a dialog box with the choice of pattern types appears (see Figure 23). The time step valid for hourly and concentration factor patterns has to be specified first. A comment can be added to this. If no other modification is required, select option "None" from the list of the pattern types before pressing the “OK” button. If a certain pattern type is selected instead, the program proceeds with lists of the current values for requested patterns. Modification of the values may now take place.

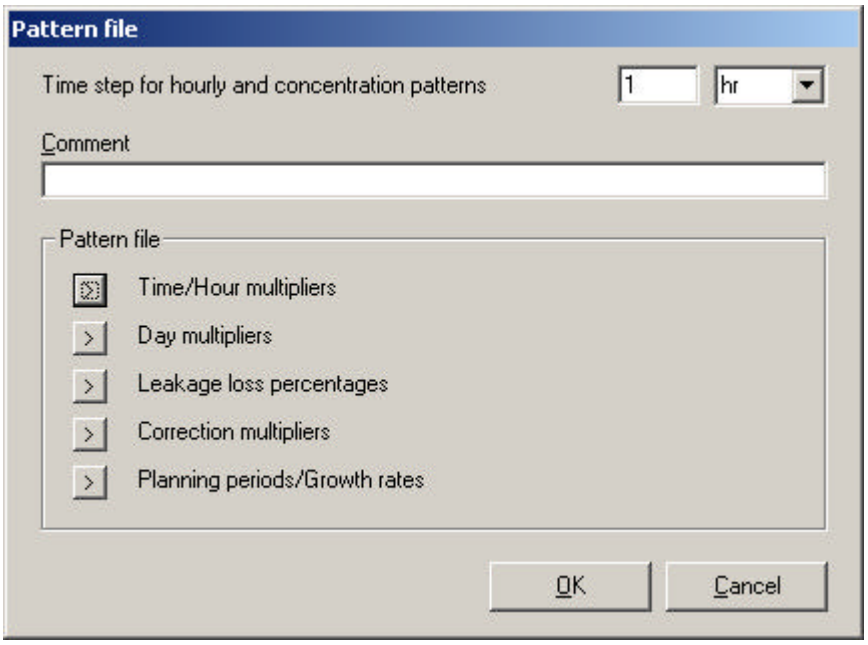


Figure 24 The pattern file menu

Time/Hour factors

By selecting this pattern type from the dialog box, a table with 24-hour patterns appears. The factors of each demand category have to be stored in the same row, while each column (except the first and the last) indicates factors at a certain moment in time. The first column contains the pattern number and the last column contains any related comments. Extending/shortening the patterns is done by filling/emptying the columns at the end of each row.

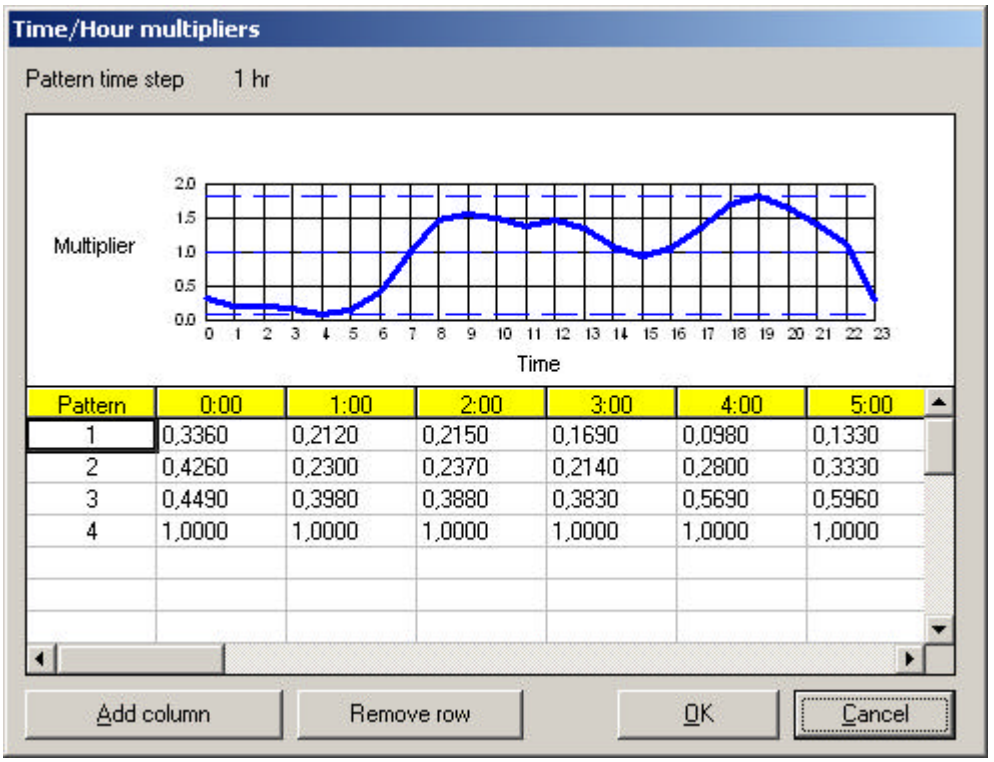


Figure 25 Time/Hour factors screen

Day factors

This option leads to a table where day factors can be specified (modified).
A maximum of seven values (one week) can be defined for each category.
A comment can be given at the end of the row.

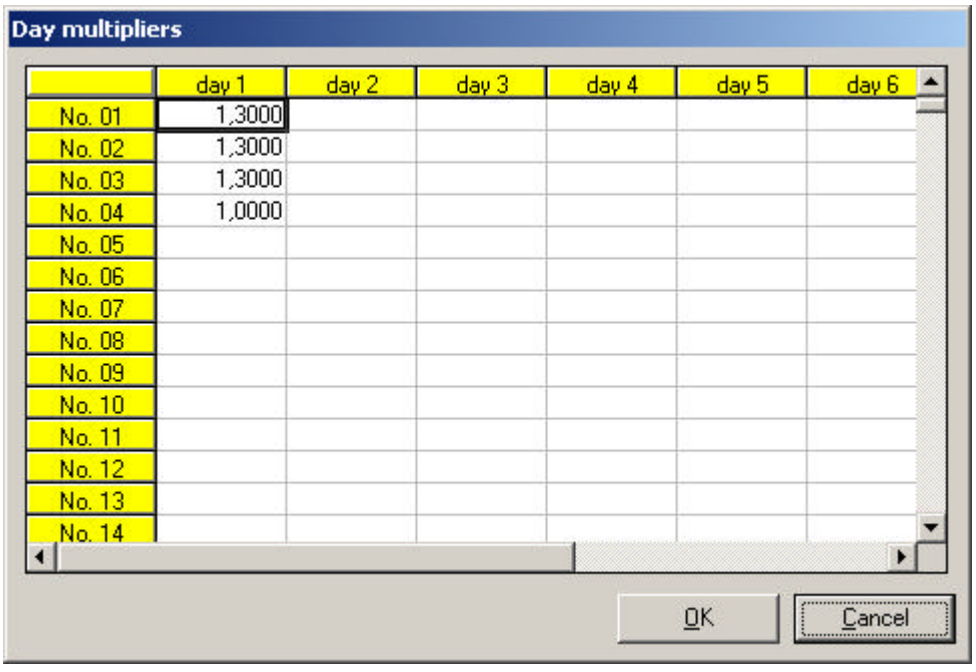


Figure 26 Day factors screen

Leakage loss percentages

The same applies as for previous tables, except that this table consists of one column only (leakage percentages for different categories).

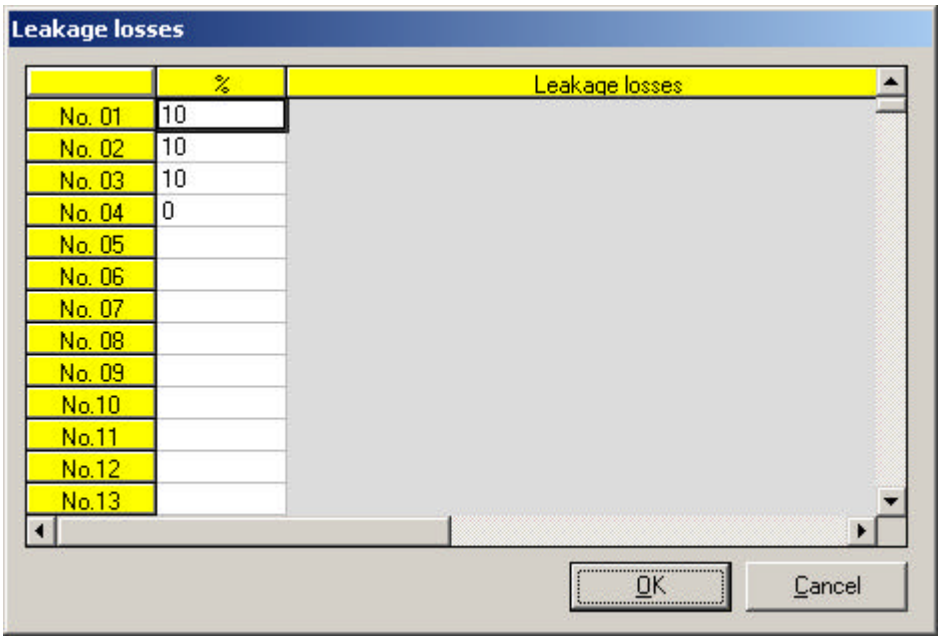


Figure 27 Leakage losses screen

Correction factors

As above.

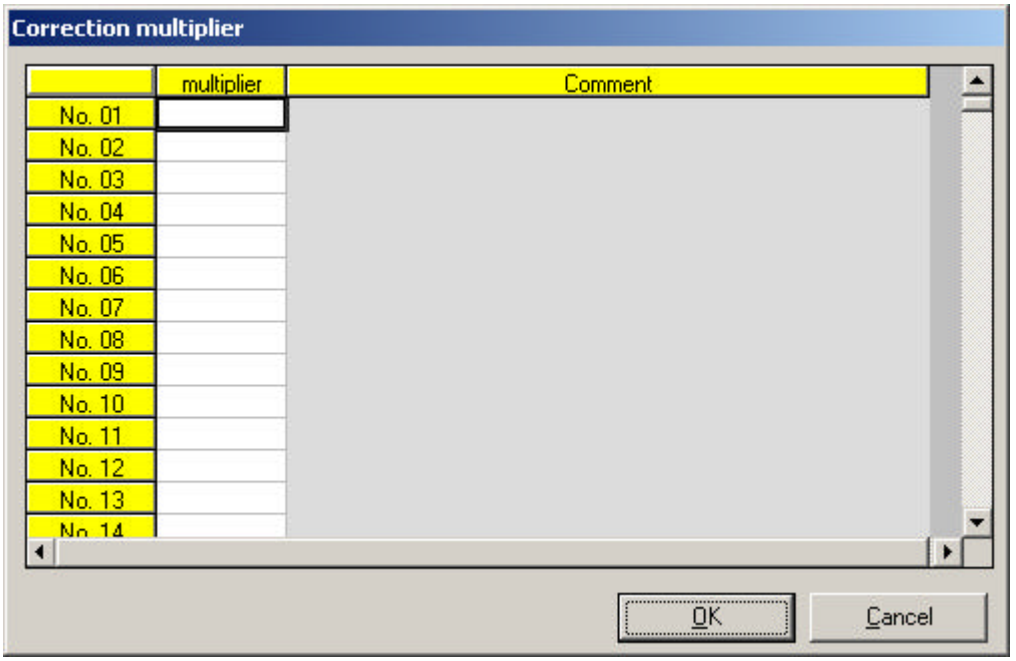


Figure 28 Correction factors screen

Planning periods / Growth rates

This option leads to the same sort of table as the previous ones, except that it consists of two parts: one for years of the periods, and another for growth percentages. If less than five periods are specified, the program stores the information in the last columns, filling the rest of the table with 0. As this is done automatically once the input is completed, no special attention to this needs to be paid by the user. A comment for each period can be added.

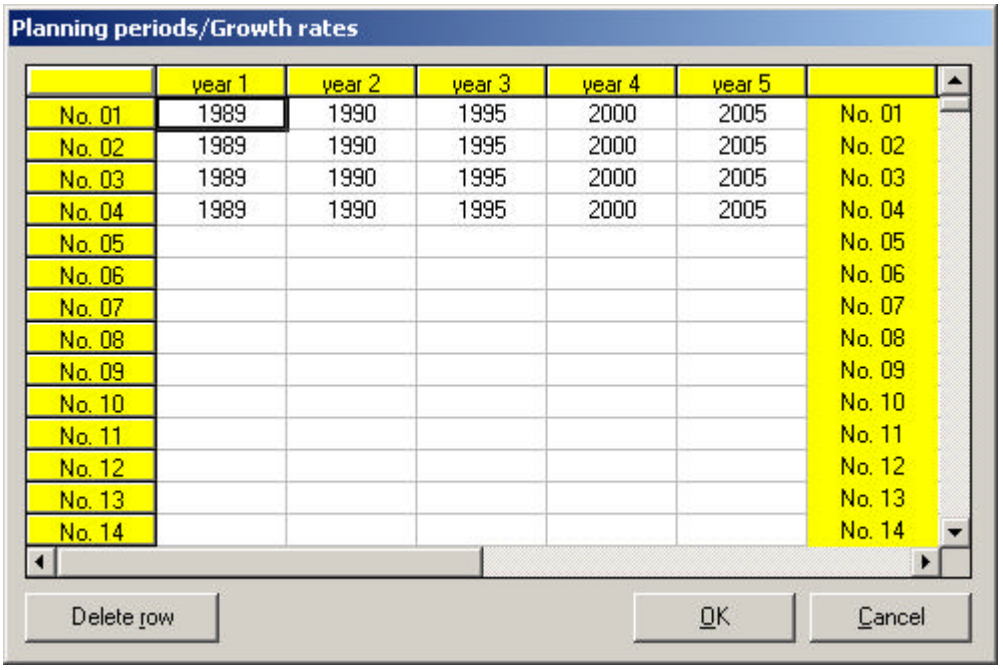


Figure 29 Planning periods/Growth rates screen: planning periods

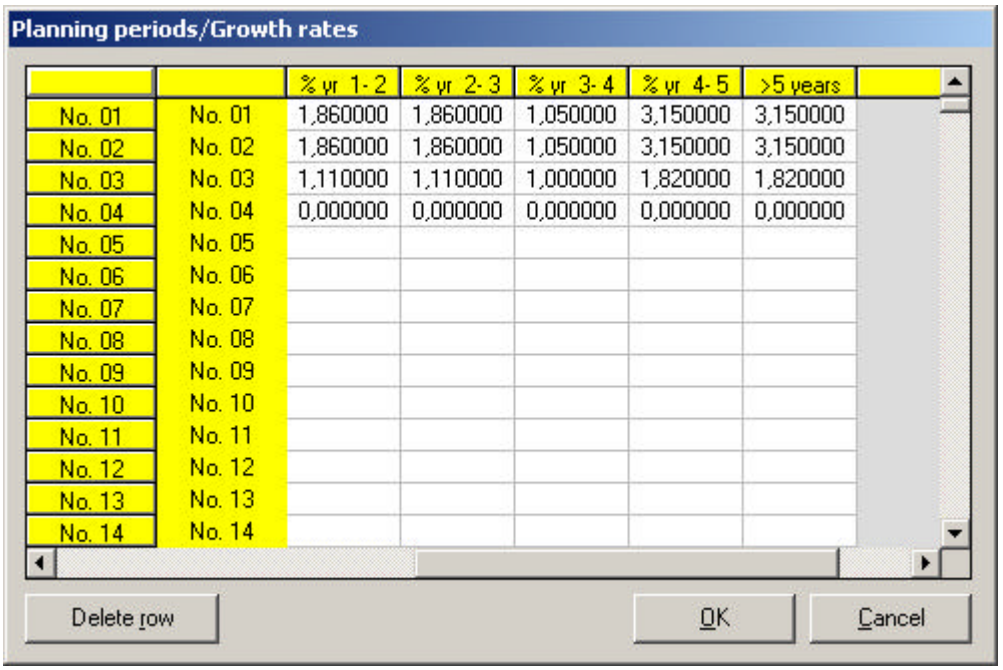


Figure 30 Planning periods/Growth rates screen: growth rates

Remarks:

- A time step has to be longer than 0.
- Leak percentages have to be equal to or greater than 0.
- Different lengths of series for different categories are allowed in case of hourly and day factors. The first time the program reads an empty cell in a row of a certain category, it assumes that this is the end of the series and proceeds with values starting from the beginning of the row. Consequently, empty cells within a series are not allowed.

Related files

Modification of the patterns is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option "File - Save input" (see section 5.4.3). This action will affect the content of the pattern file, *.PAT - all sections.

5.5.2.4 Edit Input - Controls

By running this command, a table where the control settings can be specified or modified appears on the screen (see Figures 29 and 30). The text in the table has to follow the format described in section 7.2. Additional rows (settings) can be included or the existing ones deleted from the table. The program will check the syntax of the added/modified text.

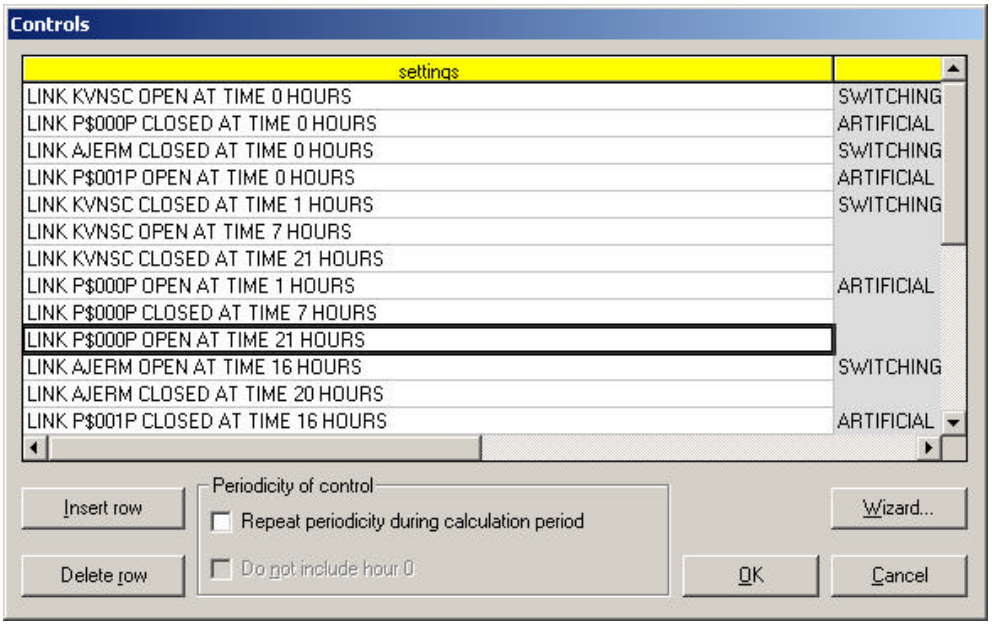


Figure 31 Controls screen, left part

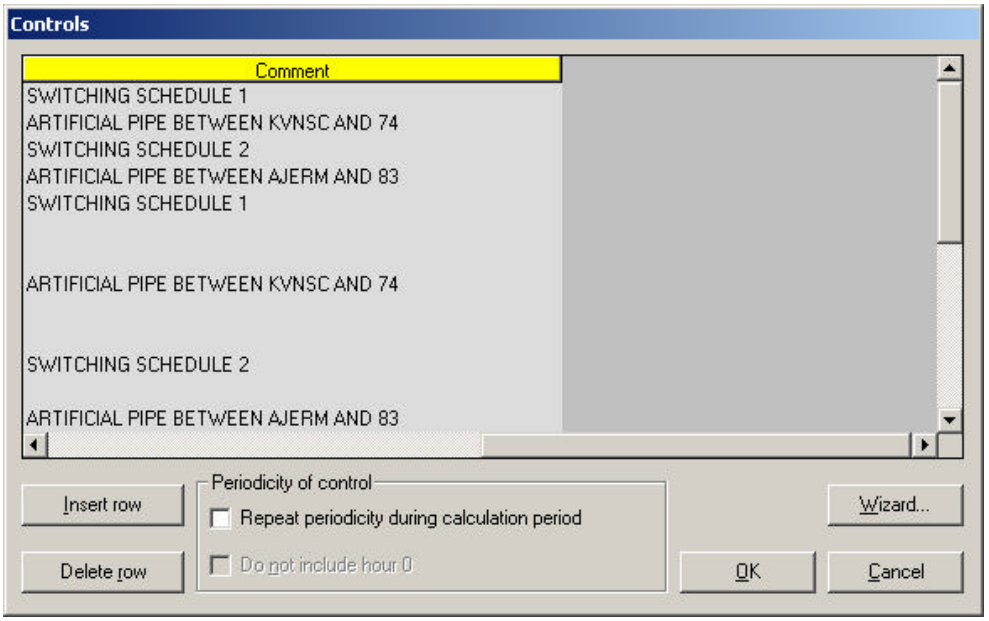


Figure 32 Controls screen, right part

Related files
Modification of the control settings is temporary: it is registered only in computer memory as long as the program is active. If it is preferred to make it permanent i.e. registered also in the input files, the menu option "File - Save input" should be chosen (see section 5.4.3). This action will affect the content of the control file, .STF - section [CONTROLS].

5.5.2.5 Edit Input - Curves

Tanks

This relation can specify the Volume-Head relation for reservoirs, where:
- the identification number has to be greater than 0;
- V, H-values have to be greater than 0.

Remarks:
- Data following an empty row will be ignored;
- Incomplete data: the missing value will be set to 0;
- The values for the V,H-relation have to be specified in ascending order.

Related files
Modification of the control settings is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option “File - Save input” (see section 5.4.3). This action will affect the content of the curves library file, .PHF - section [TDATA].

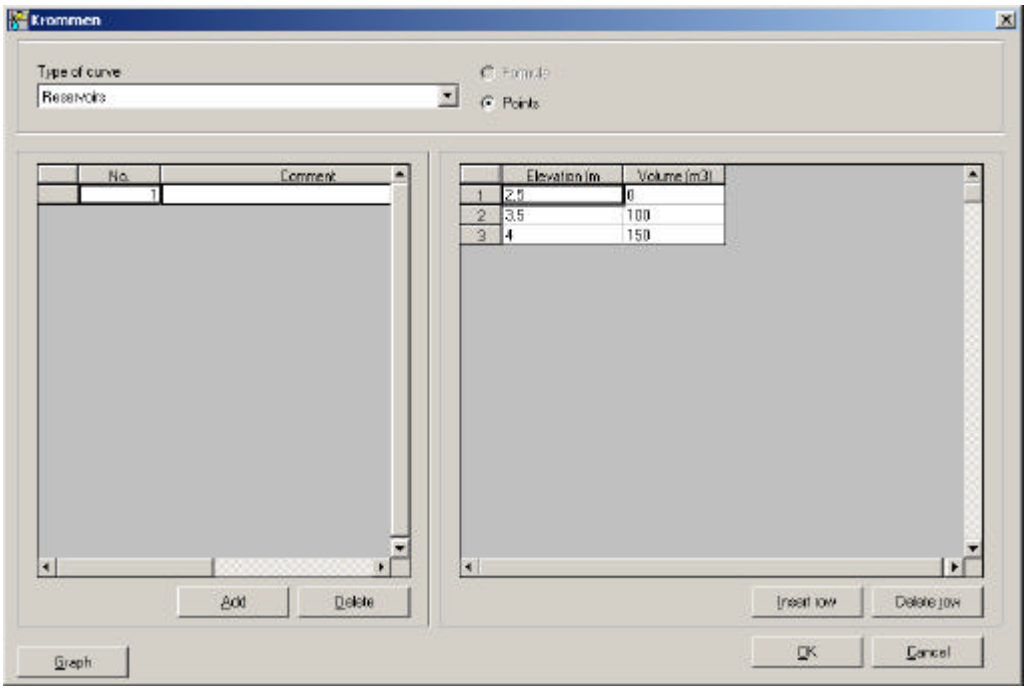


Figure 33 Edit Curves – Tank relation

Pumps / Hydrants

This relation can specify the flow - pressure jump/drop relation for pumps/ hydrants. There are two possible methods of curve definition (see section 7.14): by formula, or by polygonal line, via points. The following is valid:

- the identification number has to be greater than 0;
- the flow - pressure jump/drop values have to be greater than 0.

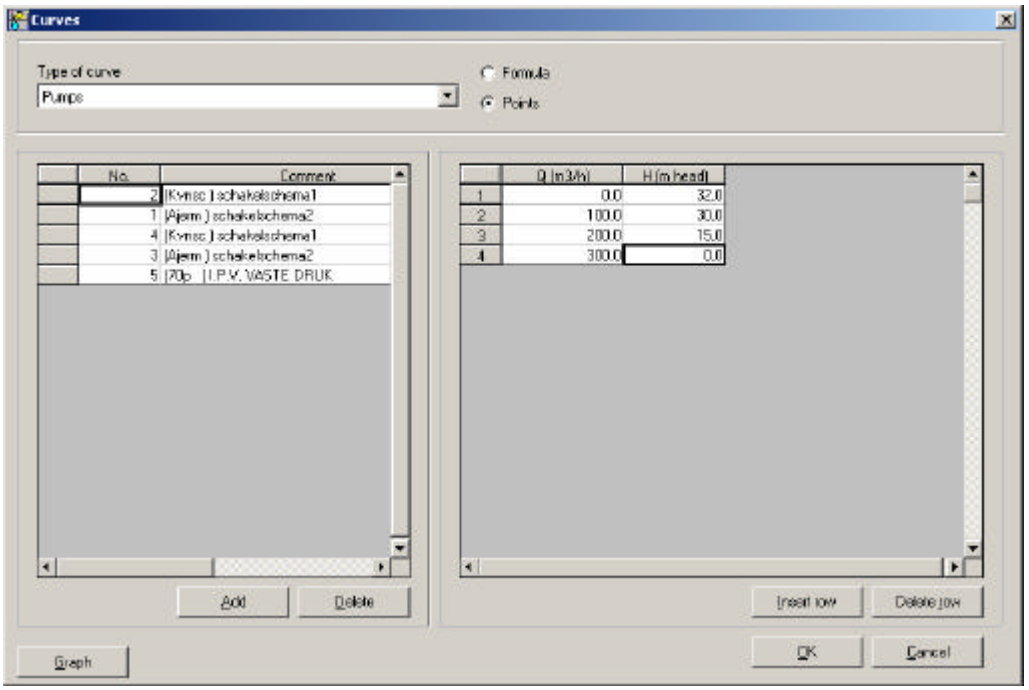


Figure 34 Edit pump curve, first screen

The menu structure of this option consists of two levels. The following should be specified in the first level (see Figure 34): the type of curve (hydrant, pump or tank) the identification number of the curve (grid on the left) and the curve type (above). Choosing the “Formula” option will change the headers in the grid on the right, where the curve values are specified. A graphic presentation of the input is also possible here.

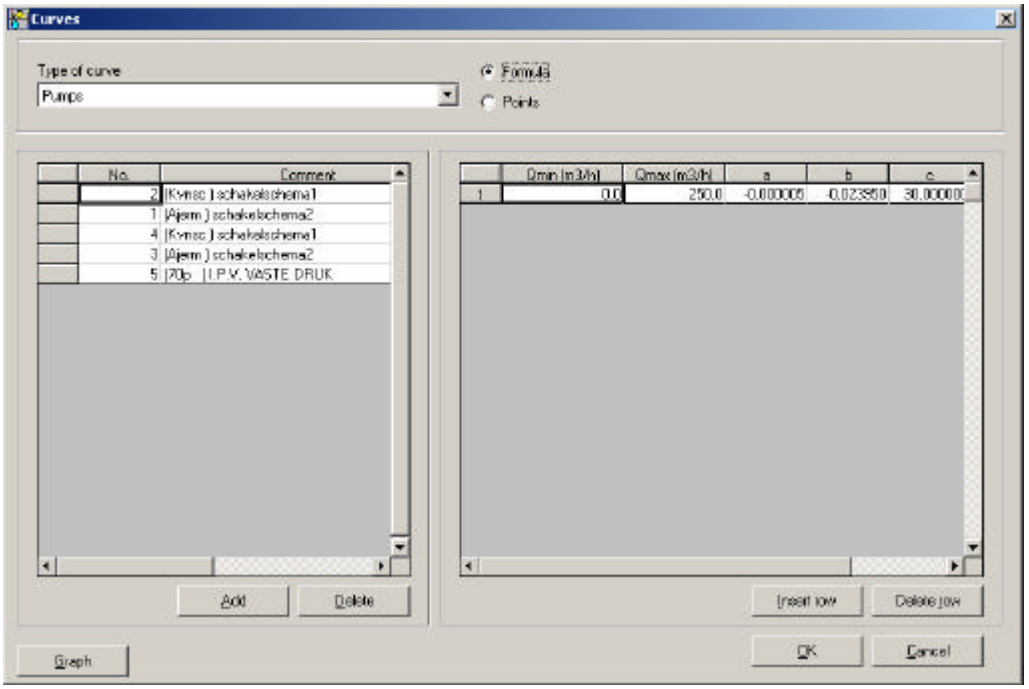


Figure 35 Edit pump curve, second screen

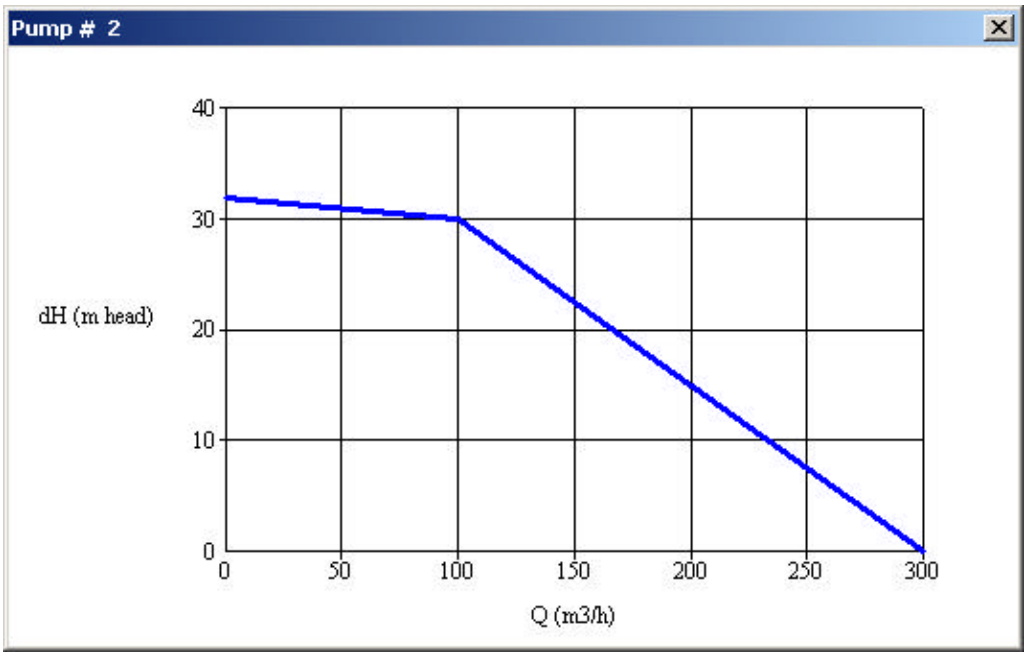


Figure 36 Pump characteristics curve, shown by clicking on ’Show’ button

Remarks

- Data following an empty row will be ignored;
- Incomplete data: the missing value will be set to 0;
- The values for pump curves have to be specified in ascending order.

Related files

Modification of the control settings is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option "File - Save input" (see section 5.4.3). This action will affect the content of the curves library file, *.PHF - sections [PUMP CURVES] and [HYDRANT CURVES].

5.5.3 Edit Input - Hydraulic Options

Various parameters used in hydraulic calculations can be modified under this option (see Figure 37):

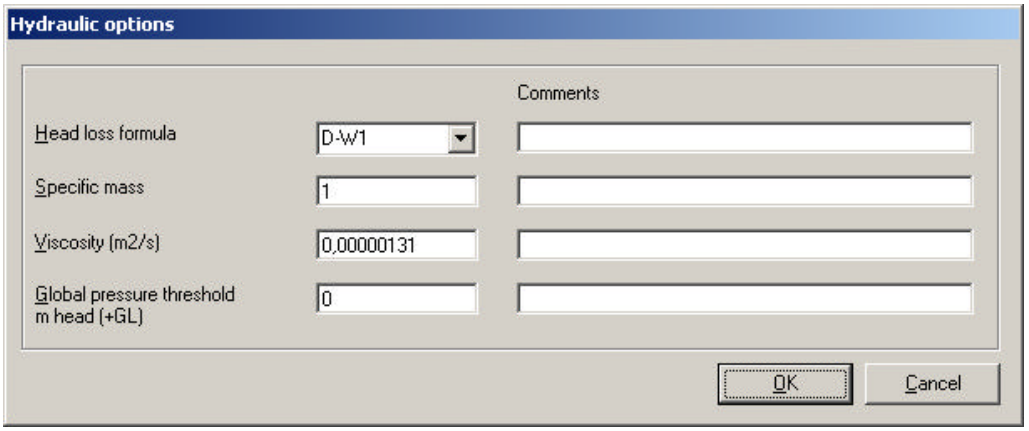


Figure 37 Hydraulic options menu

- The applied head loss formula. The roughness factor used in the input must be in accordance with the formula selected here. Options are Hazen-Williams, Darcy-Weissbach and Chézy-Manning.
- Mass density of water in the network. It is used exclusively for conversion of the pressure units.
- The kinematic viscosity. Default setting for viscosity is $1.31*10^{-6}$ m^2/s (at T = $10^{o}C$).
- The global pressure threshold (local pressure threshold is specified by the "Node" menu).

The default settings will be displayed in the dialog box if not previously changed.

Related files

Modification of the settings is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option **File**, **Save Input** (see section 5.4.3). This

action will affect the content of the control settings file, *.STF - sections [OPTIONS] and [PRESS.DEP].

5.5.3.1 Edit Input - Water Quality Options

The type of water quality calculation is displayed by this option.
The remaining menu options regarding water quality are dependent on the choice here.

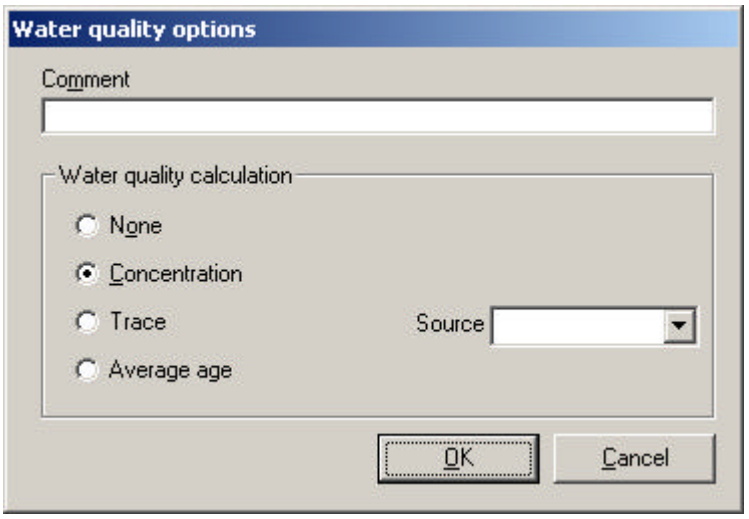


Figure 38 Water quality options menu

By selecting 'Concentration' a new dialog box appears with the general data for the concentration calculation.

These are:
- The name of the chemical substance. An arbitrary name can be given instead of the initial name "Chemical"(e.g. "Chlorine").
- The global bulk reaction coefficient. Default setting: 0 per day.
- The global wall reaction coefficient. Default setting: 0 m/day.
- The diffusion coefficient. Default setting: $1.21*10^{-9}$ m^2/day, which is diffusion at T = 20 ^{o}C.
- The maximum concentration. This is the value specified in the input section [REACTIONS] - LIMITING POTENTIAL.

By setting the percentage, the node name from which the calculation starts must be specified. This node has to be a real source, e.g. a reservoir node.

Related files
Modification of the water quality options is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option **File**, **Save Input** (see section 5.4.3). This action will affect the content of the water quality file, *.WQI - sections [QOPTIONS] and [REACTIONS].

5.5.3.2 Modify input - Concentration patterns

In concentration patterns, a similar menu appears as for hourly factors (see section 5.5.2.3). The same procedure can also be applied in this case. Note that the time interval of the demand and concentration pattern is the same.

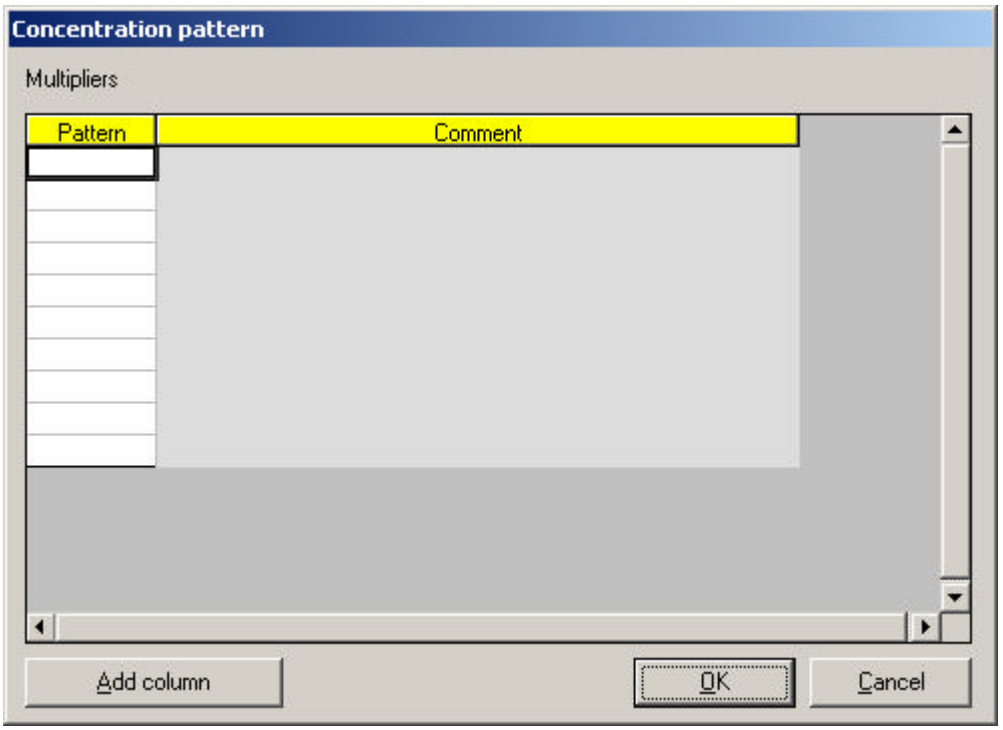


Figure 39 Concentration pattern menu

Related files
Modification of the concentration patterns is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option **File**, **Save Input** (see section 5.4.3). This action will affect the content of the water quality file, *.WQI - section [QPATTERNS].

5.5.3.3 Edit Input - Contour

This command gives a list of the points describing the contour lines, with the possibility of modifying or deleting these, or adding new points. One contour line is separated from the others in the list by the asterisk character.

Related files
Modification of the contour lines is temporary: it is registered only in computer memory as long as the program is active. If you prefer to make it permanent i.e. registered also in the input files, select the menu option **File**, **Save Input** (see section 5.4.3). This action will affect the content of the contour file, *.CON.

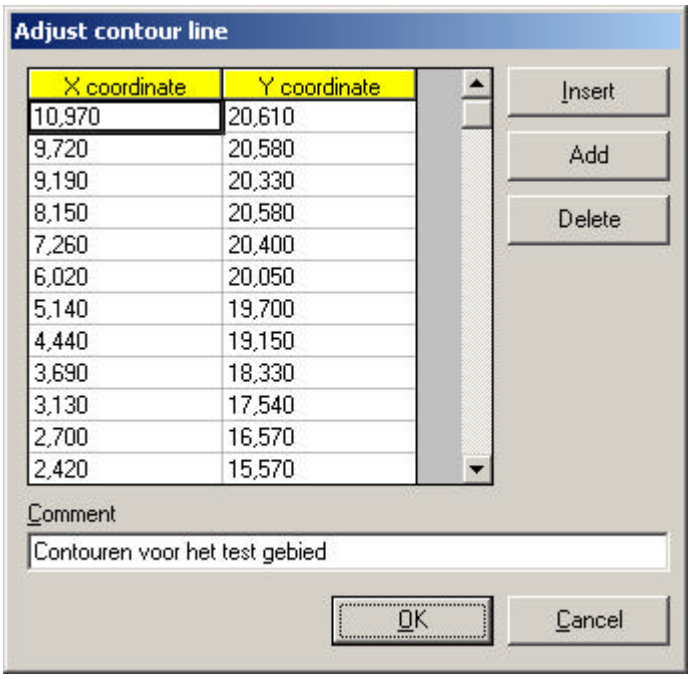


Figure 40 Adjust contour line menu

5.5.4 Copy

This command copies text or figures (bitmap) from the active window to the Windows clipboard, from where it can be used (pasted) in other Windows applications (e.g. the standard Windows text processor Notepad). The copied data remain on the clipboard temporarily and has to be saved for further use. This is done separately via the clipboard (.CLP files). The content of a clipboard file can be used later. It can also be printed from various applications.

The command is accessible from:
- map presentation
- diagram
- table
- viewer.

5.6 Main menu: option “Calculate (F5)”

5.6.1 Standard

This option allows the calculation run parameters to be set, i.e. initialisation of the calculation.

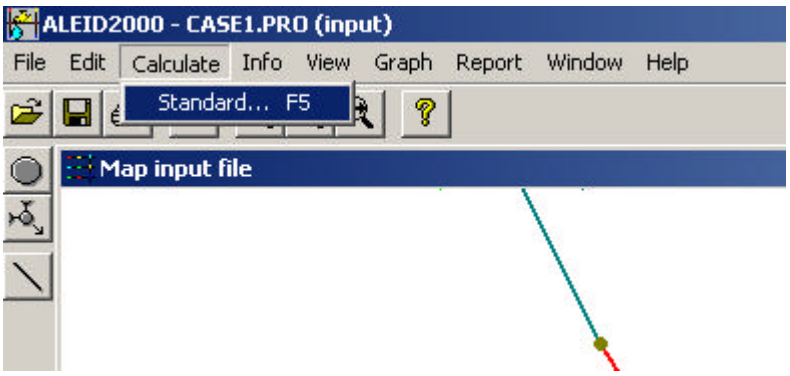


Figure 41 Calculate option

A dialog box appears with questions about the hydraulic calculation, water quality calculation and the format of input and output (see Figures 42-44). The values specified are then stored in the file ALCALC.INI (see section 4.13). The following is the description of the format of the ALCALC.INI file in relation to the content of the corresponding dialog box in the program.

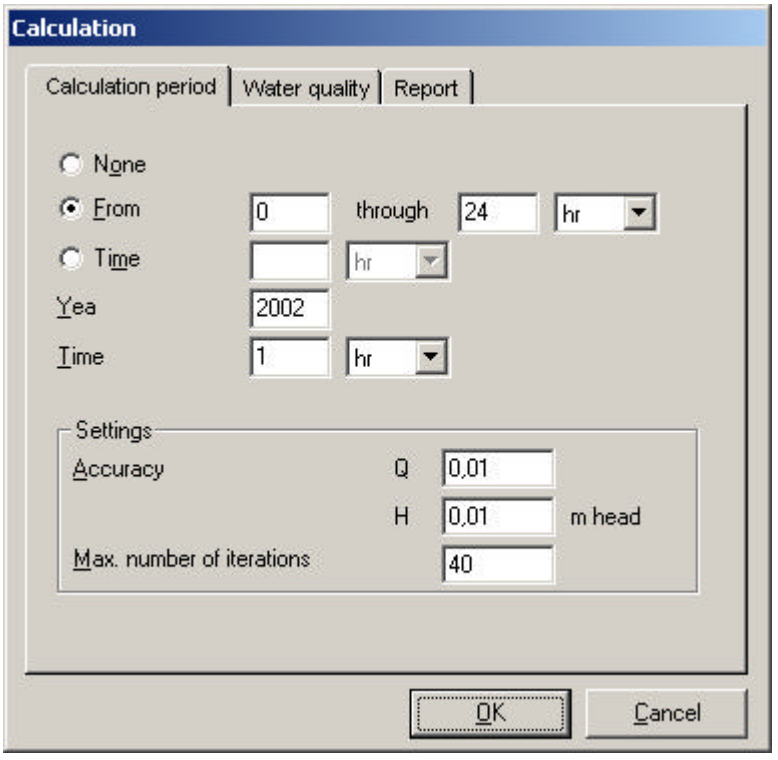


Figure 42 Calculate screen; ‘Calculation period’ tab

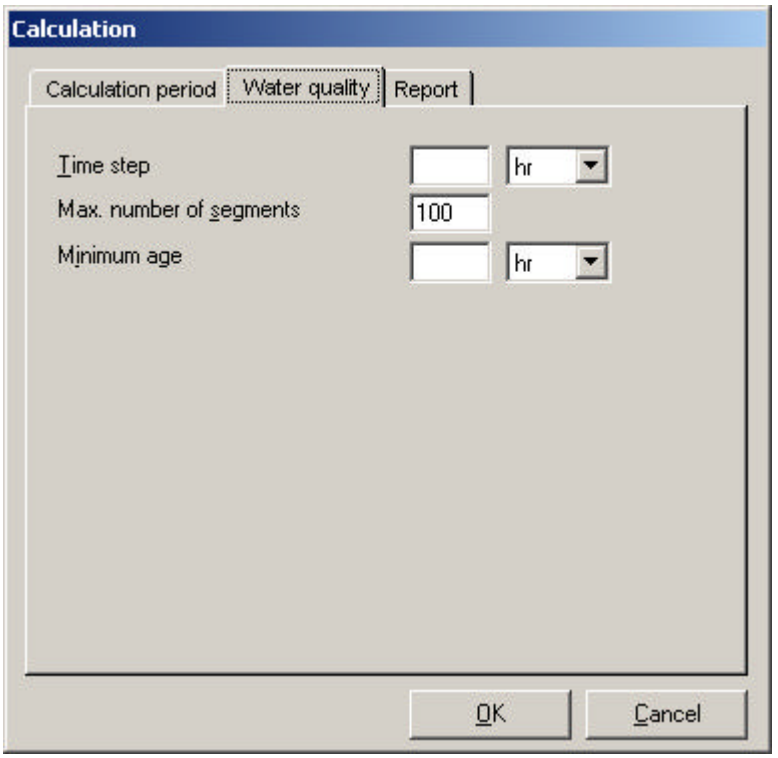


Figure 43 Calculate screen; 'Water quality' tab

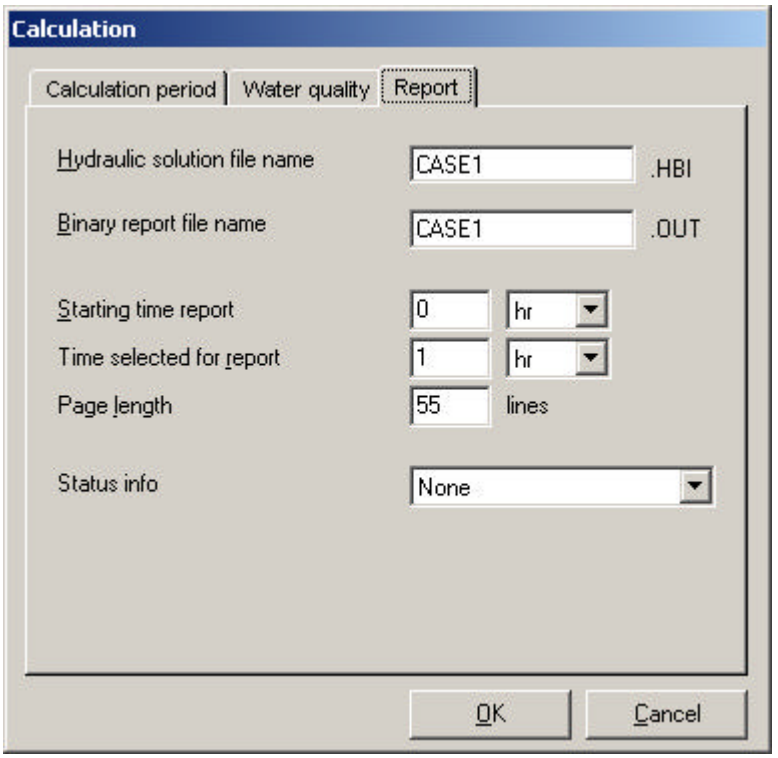


Figure 44 Calculate screen; 'Report' tab

Calculation period - section [GENERAL]

- Simulation interval
  Choice from:
  * None (already existing file with hydraulic solution) --
  * From (t1) to (t2) START t1, STOP t2
  * snapshot at (tx) START tx, STOP tx
- Time step between two calculations TIMESTEP
- Calculation year YEAR
- Calculation accuracy for flow and heads ACCURACY
- Maximum number of iterations TRIALS

Water quality - section [QTIMES]

- Time step between two calculations QUALITY TIMESTEP
- Maximum number of segments SEGMENTS
- Minimum travel time MINIMUM TRAVELTIME

Report - sections [SOLUTION] and [REPORT]

- Name of interim binary file with hydraulic solution:
  [SOLUTION] - HYDRAULICS SAVE or USE, depending on purpose
- Name of binary report file [REPORT] - BFILE
- Start time in the report [REPORT] - REPORT START
- Time step in the report [REPORT] - REPORT TIMESTEP
- Page size in the report [REPORT] - PAGESIZE
- Status info [REPORT] - STATUS

By modification of the ALCALC.INI file, the previous version becomes overwritten. The program proceeds by calling the calculation module ALCALC.EXE.

There are several possibilities in performing a calculation:

- as a hydraulic calculation only. If the water quality file, *.WQI is not present, or the “None” water quality calculation is specified, that calculation will not be performed. The basic results of the hydraulic calculation can be stored in the interim binary file with extension .HBI. Based on this information, a water quality calculation can be done later if necessary.
- as a water quality calculation only. If “None” calculation period is selected in the dialog box, the program reads the results of hydraulic calculations made earlier as stored in the .HBI file specified in the cell “Hydraulic solution”. A water quality calculation will be then executed based on the statements in the .WQI file.
- If for the calculation period “None” is selected in the dialog box (ALCALC.I-NI), and “None” water quality calculation is requested, the program will create the binary output file (*.OUT) based on the project file and the file with hydraulic calculations specified in the dialog box.
- Both calculations: the complete calculation will be executed if a water quality option in the .WQI file and a simulation interval in the dialog box

(ALCALC.INI) are specified. If a file name in the “Hydraulic solution” cell is given as well, the results of the hydraulic calculation will be stored there for possible later use in other water quality calculations.

After the calculation is executed, the program creates a binary file with the output results (*.OUT) and an error-/status report file with name $. The content of this file can be inspected via the “Output summary” option. The $ file can also be accessed in the viewer or by the editor at any moment.

If the calculation is completed successfully, the content of the .OUT file will be immediately used for presentation of the pressure situation in the network (in colours), unless specified differently in the ALEID2000.INI file. The remaining results can also be shown in the map by using the options available in the window.

Remarks

- Presentation of the output removes the input information from the program memory. If a new calculation is necessary or input data modification is requested, the project file has to be retrieved again.
- If no .WQI file is specified, the water quality options cannot be accessed.
- If one or more water quality parameters are not specified, the program assumes the default settings (see section 7.18). These are not given for the time step and the minimum travel time, being calculated by the program itself.
- The water quality status in the content output (the file $) should be checked after every water quality calculation. If many links exceed the specified segmentation (in the last column) and the calculation results show unexpected jumps, the calculation should be repeated with a larger number of segments. NOTE: The segmentation is inversely proportional to the water quality time step (default setting: 1/10 of the time step for hydraulic calculation). Hence, the smaller the time step is, the more segments will be required.

5.7 Main menu: option “Info”

5.7.1 All Nodes & Pipes

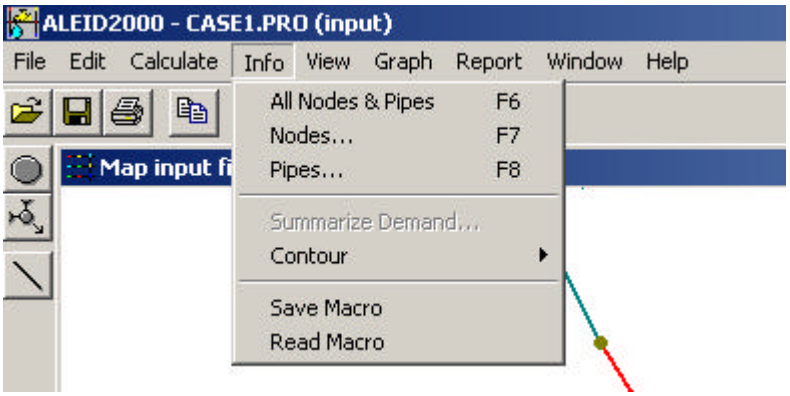


Figure 45 Info selection window

Under this option, a list of available network parameters is shown on screen. Selected parameters will be displayed (in numbers) on the map for all nodes and/or links. Several parameters can be shown simultaneously; however, some combinations cause overlapping of numbers, which then become barely legible. The following parameters can be selected (see Figure 46):

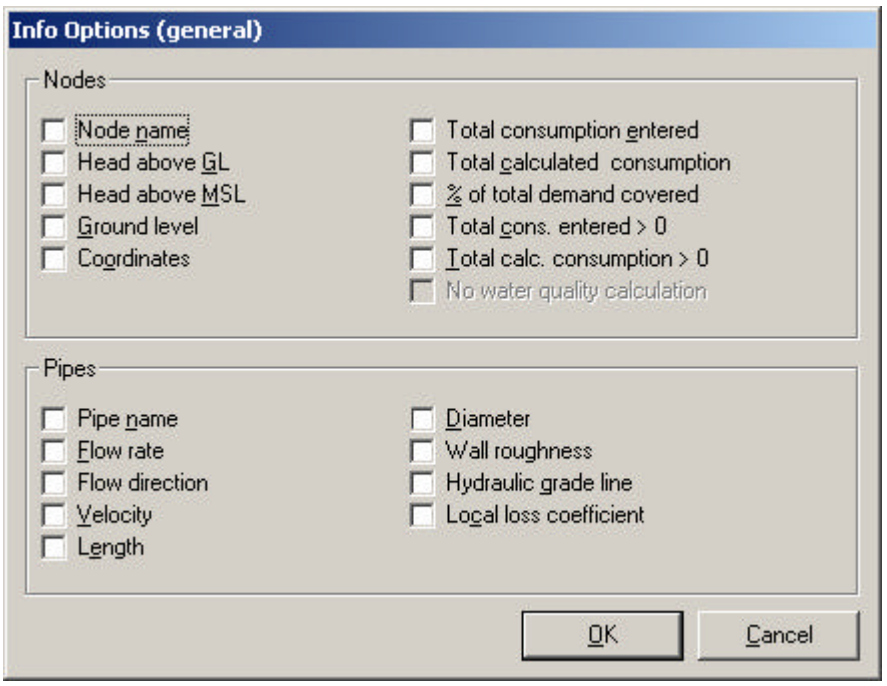


Figure 46 Info Options window

Nodes:

-	Node name	I/O
-	Head above GL	O
-	Head above MSL	O
-	Ground level	I/O
-	Co-ordinates	I/O
-	Water quality	W
-	Total consumption entered	O
-	Total calculated consumption	O
-	% of total demand covered	O
-	Total consumption entered > 0	O
-	Total calculated consumption > 0	O

Pipes:

-	Pipe name	I/O
-	Flow rate	O
-	Flow direction	O
-	Velocity	O
-	Length	I/O
-	Diameter	I/O
-	Wall roughness	I/O
-	Hydraulic grade line	O
-	Local loss coefficient	I/O

I = available if input information is retrieved;
O = available if output information is retrieved;
W = available if water quality calculation was performed.

Parameters that are not accessible for presentation at a particular moment will be 'greyed out'.

5.7.2 Nodes

This menu option allows the display of certain parameters for a number of nodes in the system. These nodes can be selected by clicking the mouse directly on the map or by making a selection from the list of all existing nodes (see Figure 47).

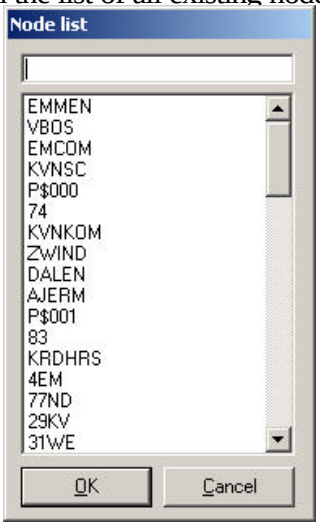


Figure 47 Example of node list

The mouse selection ends by clicking the right mouse button. The choice of available parameters is the same as in the node section of the **All Nodes & Pipes** command.

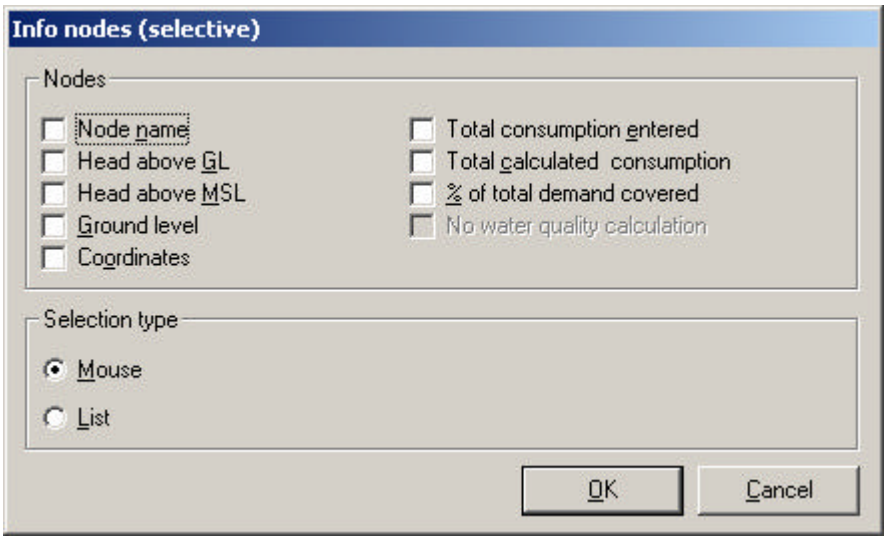


Figure 48 Info nodes window

5.7.3 Pipes

This menu option allows the display of certain parameters for a number of links in the system. These links can be selected by clicking the mouse directly on the map or by making a choice from the list of all existing links. The mouse selection ends by clicking the right mouse button. The choice of available parameters is the same as for the link section in the **All Nodes & Pipes** command.

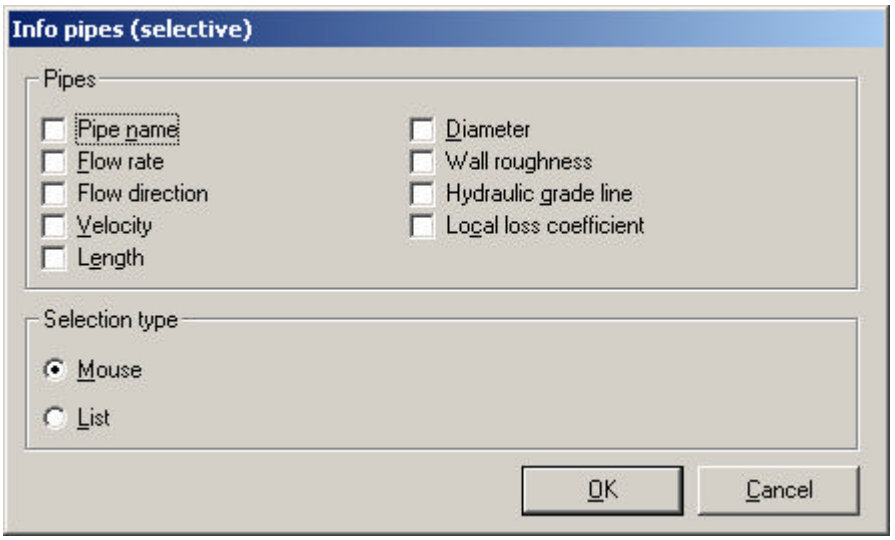


Figure 49 Info pipes window

In order to apply the model for the development of flushing programs, an extra feature added here is the presentation of the residence time in a link. This option is available only during calculations of one steady state (snapshot). Simulation of residence time over a certain period is not (yet) included in the program, although conclusions can be drawn from a number of steady state calculations.

5.7.4 Summarize demand

This command summarises the total calculated demand for a number of selected nodes. It can be activated only when an output file has been selected. The selection ends by clicking the right mouse button.

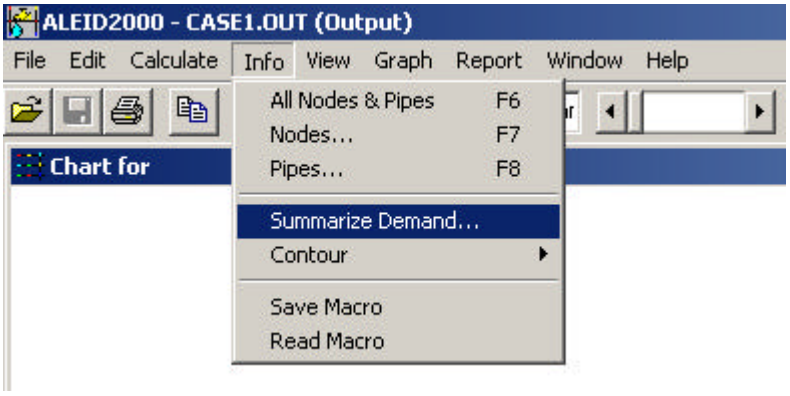


Figure 50 Summarize Demand selection

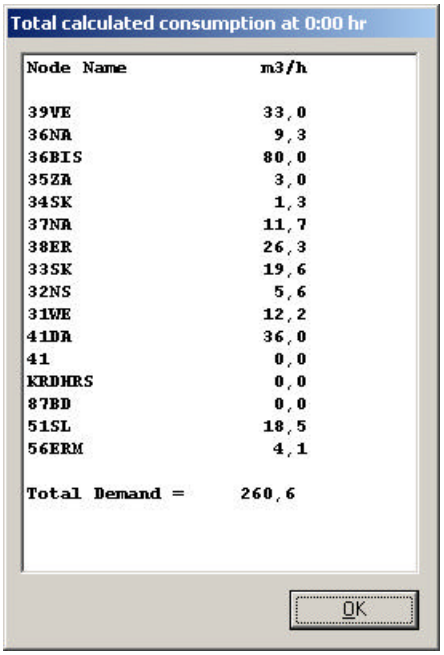


Figure 51 Resulting window

5.7.5 Contour

This applies or removes the contour lines from the map. This is done by a small dialog box, with the options **On** and **Off**. The contour file (*.CON) is retrieved together with the input or output; however, it is not a compulsory part of the model information.

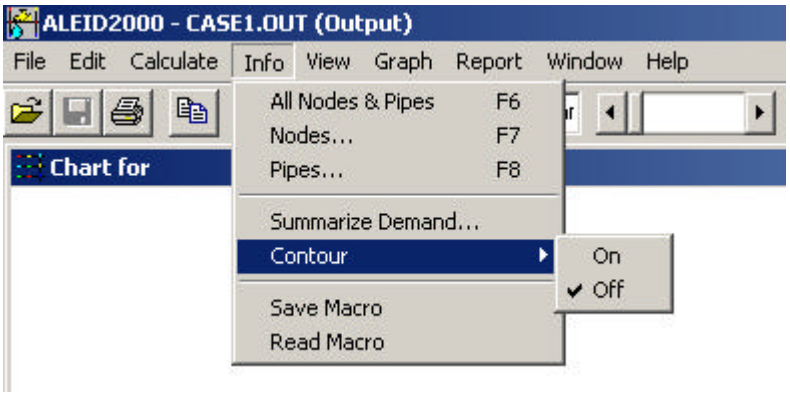


Figure 52 Contour selection options

See paragraphs 4.14 and 5.7.5 for details on contour files.

5.7.6 Save macro

This command saves the current screen settings in a macro file. These are:

- parameter colour settings
- written information
- legend settings
- aspect ratio (screen proportion)
- zoom-ins.

The settings related to the model input information will be stored in an input macro (*.MCI), whereas the settings related to the model output information will be stored in an output macro (*.MCO). Both types of macros can be applied only for their corresponding input/output screens.

5.7.7 Read macro

This calls a macro file, *.MCI or *.MCO, with previously saved screen settings (colours, text, zoom-ins etc.).

5.8 Main menu: option "View"

5.8.1 Zoom in

By this command a part of the network will be displayed at a larger scale. Selection of the rectangular area is done by dragging the mouse from lower left to upper right corner. After the mouse button is released, the selected box will be shown over the entire screen.

NOTE: This option works with a mouse only.

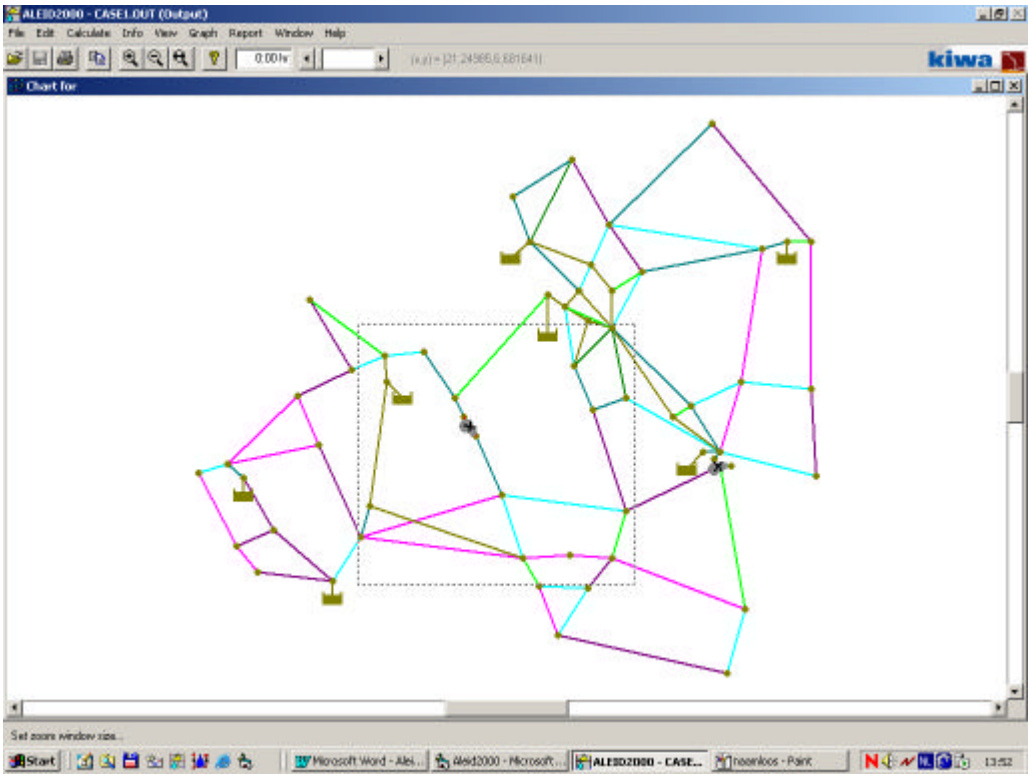


Figure 53 Selection of area to zoom in to

An alternative option for zooming in on an area is provided through buttons at the top of the screen:

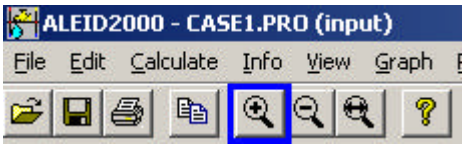


Figure 54 Alternative zoom options: zoom in to selected area

5.8.2 Zoom out

This command restores a previous zoom-in on the screen. An alternative is again provided through a button at the top of the screen.

5.8.3 Zoom all

With this command the map will be shown in the original size.

Figure 55 Alternative zoom option: back to full network

5.8.4 Clear Map

All information applied to the map will be deleted by this command, so that only the original network layout is shown once more.

5.8.5 Colour coding

This option displays a menu from which a certain parameter can be selected for graphic presentation in colours. The setting remains saved in the ALEID2000.INI file after the program is terminated. One node and one pipe parameter can be selected for simultaneous presentation. In a number of cases this can be the same parameter for both nodes and pipes. Depending on the type of retrieved information (I= input, O= output, W= water quality output) the following parameters can be chosen:

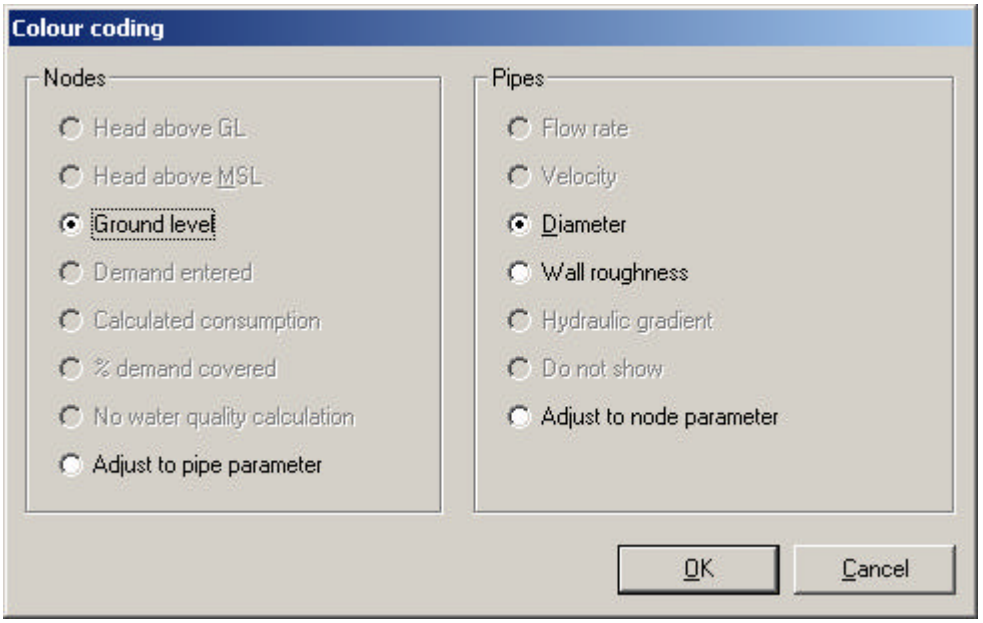


Figure 56 Colour coding settings menu

Nodes:

- Head above GL O
- Head above MSL O
- Ground level I/O
- Demand entered O
- Calculated consumption O
- % demand covered O
- Water quality (Concentration) W
- Adjust to pipe parameter I/O

Pipes:

-	Flow rate	O
-	Velocity	O
-	Diameter	I/O
-	Wall roughness	I/O
-	Hydraulic gradient	O
-	Do not show	I/O
-	Adjust to node parameter	I/O

5.8.6 Show Legend

This option displays or removes the legend(s) from the screen. Two legends are commonly displayed: one for node and one for link data. If the same parameter is chosen both for nodes and links, only one legend will appear. The legends can be moved to another position by dragging the mouse after placing the cursor inside the legend box.

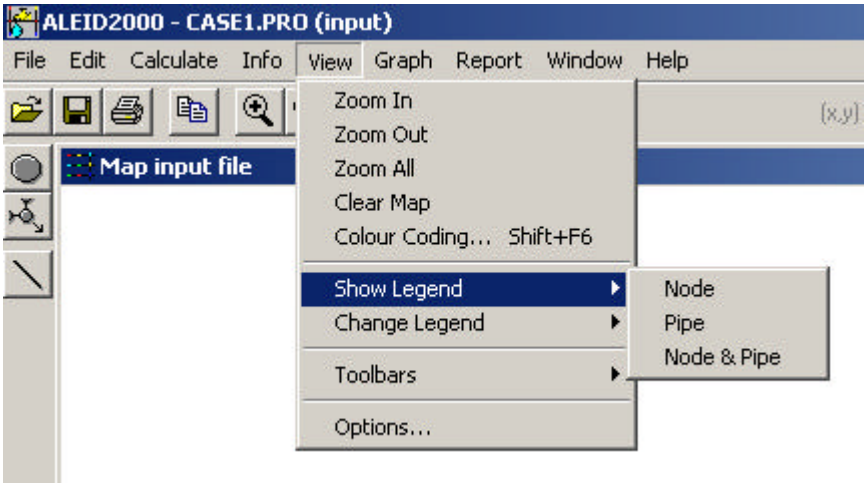


Figure 57 Show/Change legend menu

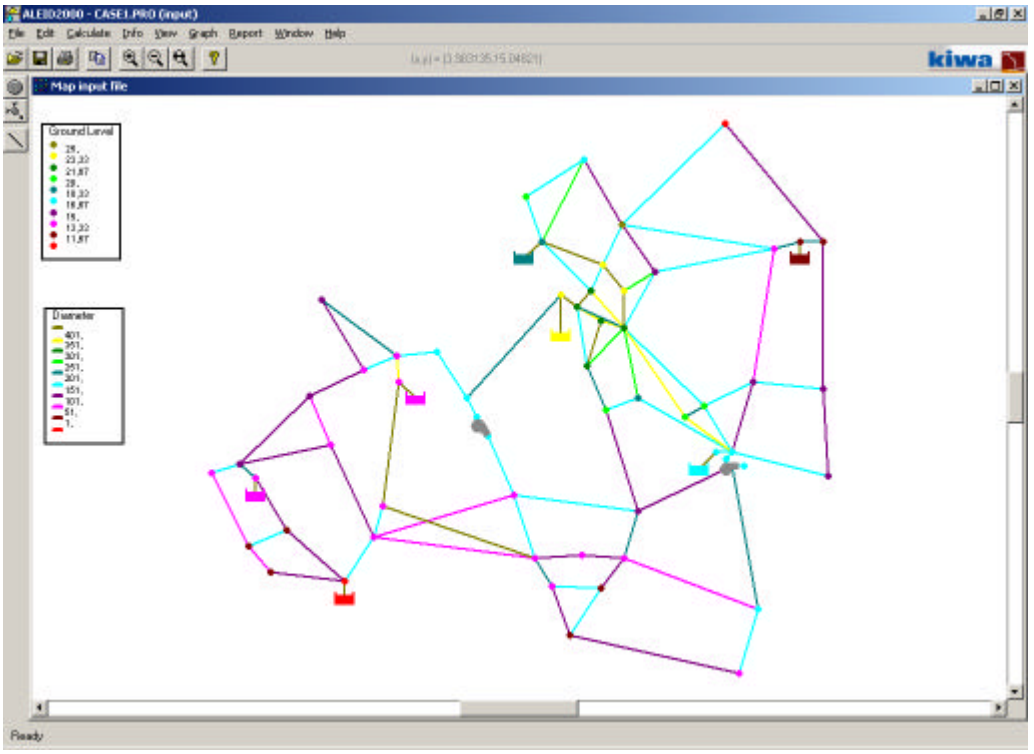


Figure 58 Legends shown on screen (2 boxes at the left)

5.8.7 Change Legend

This command modifies the ranges and colours for a selected legend (see Figure 57). The values can be modified in two ways:

- individually, changing values one by one in the left column;
- starting with a minimum value and an interval, specified in the right hand column. This choice automatically updates the values in the left-hand column.

Clicking the mouse on the selected colour cell in the dialog box modifies the colours. A standard Windows menu with available choices will appear. Not all of the offered colours can be used for screen presentation, depending on the screen and video memory/card type.

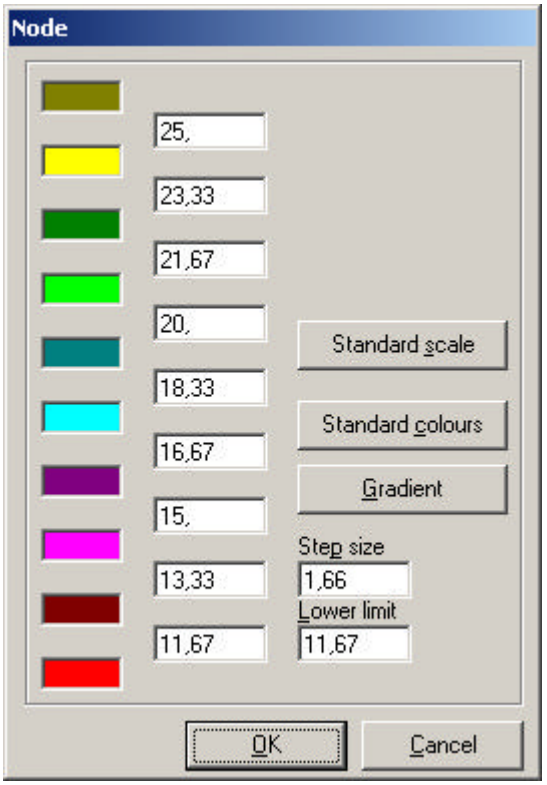


Figure 59 Change Legend dialog box

The settings remain saved in the ALEID2000.INI file after the program is terminated and are re-installed during later use.

5.8.8 Toolbars

This option has 2 sub-options: **Standard** and **Toolbox**, which allow the buttons at the top and to the left of the screen to be switched on or off.

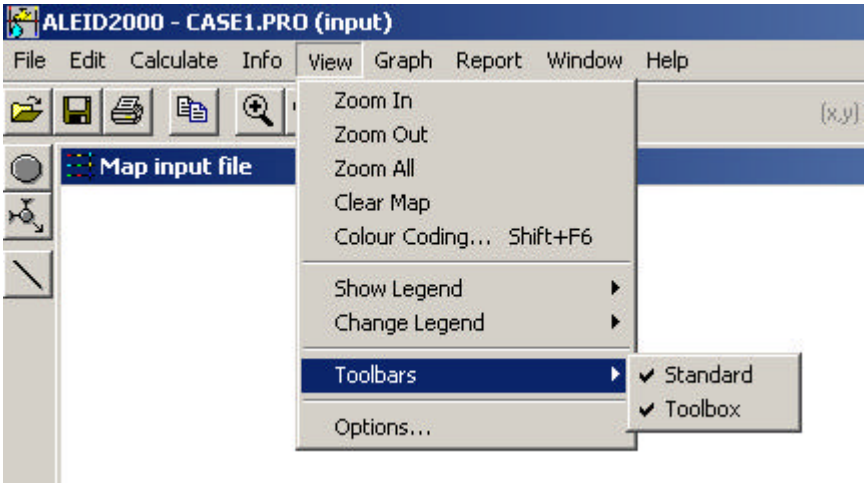


Figure 60 Toolbars setting box

5.8.9 Options

This option allows modification of standard presentation parameters:

- Node presentation i.e. the circle size;
  Available choices:
  * None
  * Small
  * Large
- Link presentation i.e. the line thickness;
  Available choices:
  * Thin
  * Thick
  * Extra thick (for larger screens)
- Background colour;
  Available choices:
  * Black
  * White
  * Grey

- Highlight (node selection);
  Available choices:
  * Square
  * Point

- Global;
  Available settings:
  * Aspect ratio (default setting 1.0). This is a parameter that “stretches” (if >1) or “compresses” (if <1) the graph along the horizontal axis.
  * Symbols (tanks, valves etc.) on or off.

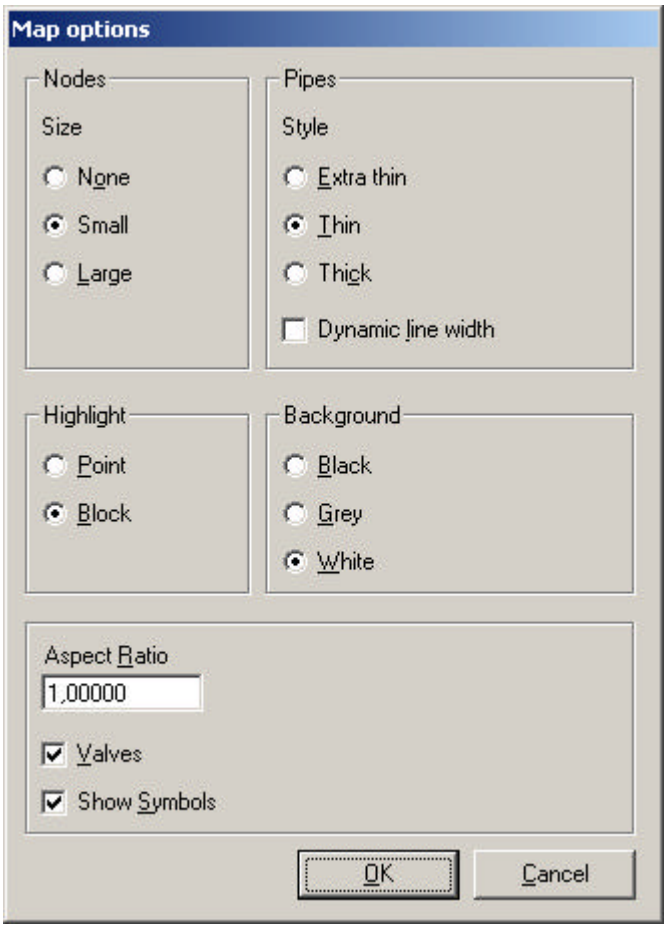


Figure 61 Map options setting window

5.9 Main menu: option "Graph"

5.9.1 Time Series Node

Under this option, the progression of a certain nodal parameter throughout a specified period can be displayed (see Figure 60). A maximum of five curves (nodes) can be shown in one diagram. The node selection can be done on the map (by mouse, the right button ends the selection) as well as from the list of the nodes (the "OK"-button ends the selection). A legend will be added to the diagram depending on the choice of nodes. Moreover a minimum and maximum value will be indicated for each series.

The graph includes the simulation period starting from a pre-selected point in time. Obviously, the diagram cannot be generated if that moment is the end time of the simulation run. For comparison of the calculation results with real system operation, a file, *.DAT, with measured data can be added via the option **Graph**, **Options** (see below).

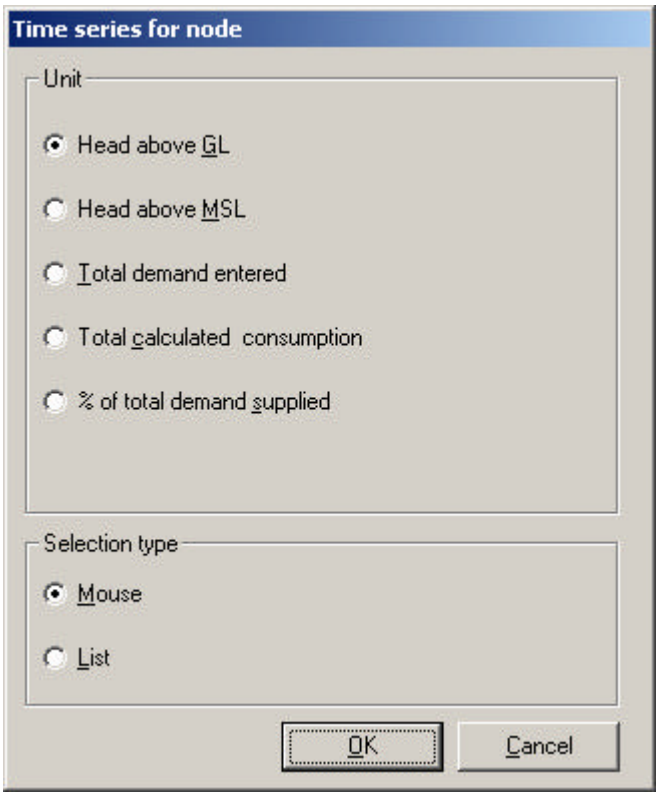


Figure 62 Setting parameters for time series for node

Examples of the resulting output for pressure head and consumption at a particular node are given in Figures 63 and 64, respectively.

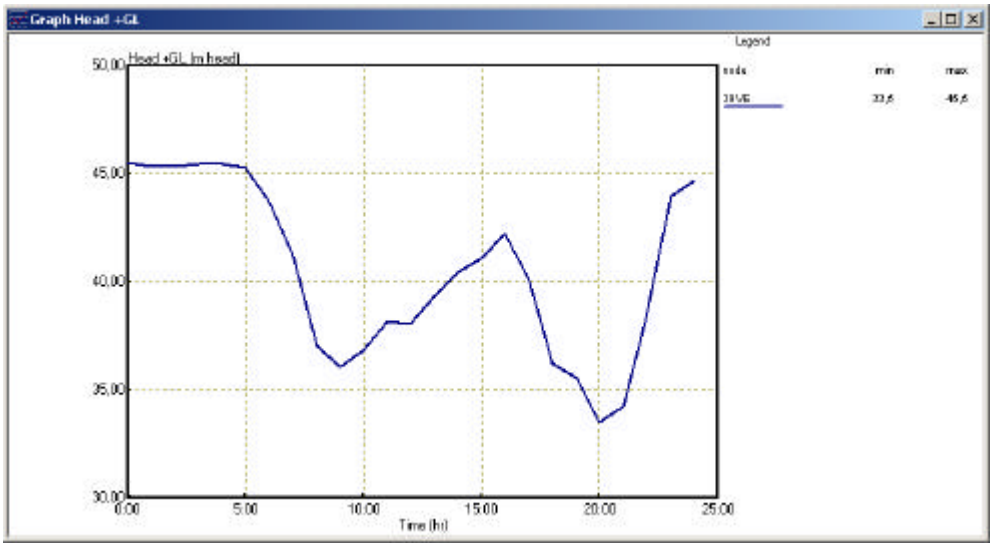


Figure 63 Time series for node: pressure head

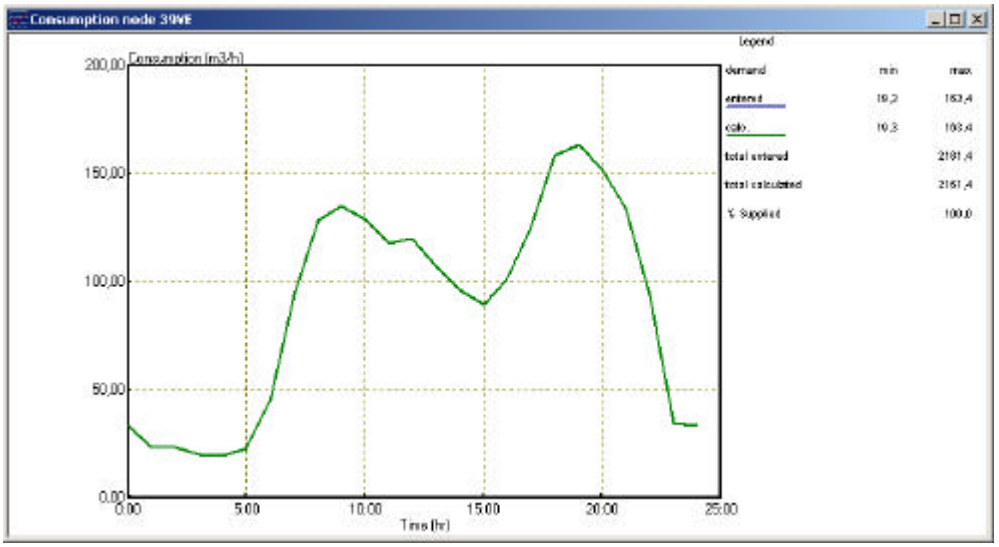


Figure 64 Time series for node: consumption

5.9.2 Time series pipe

Under this option, the progression of a certain link parameter throughout a certain period can be displayed. A maximum of five curves (links) can be shown in one diagram. The link selection can be done on the map (by mouse, the right button ends the selection) as well as from the list of the links (the "OK"-button ends the selection). A legend will be added to the diagram depending on the choice of links. Moreover a minimum and maximum value will be indicated for each series.

For comparison of the calculation results with real system operation, a file (*.DAT) with measured data can be added via the option **Graph**, **Options** (see below).

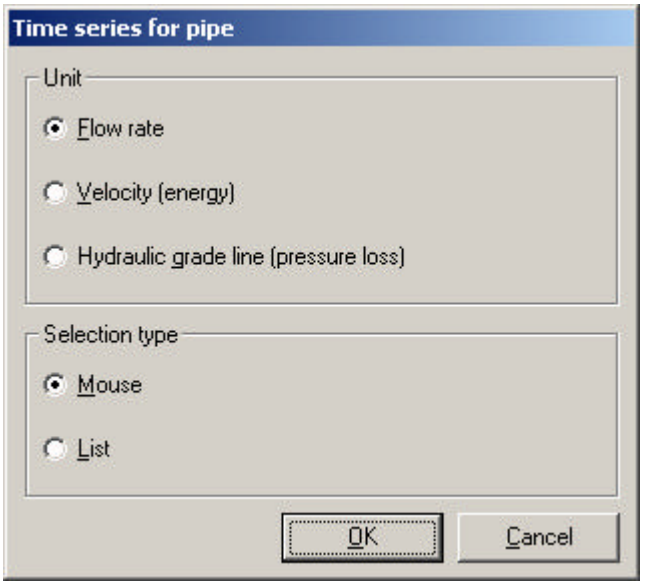


Figure 65 Time series for pipe: flow rate selected

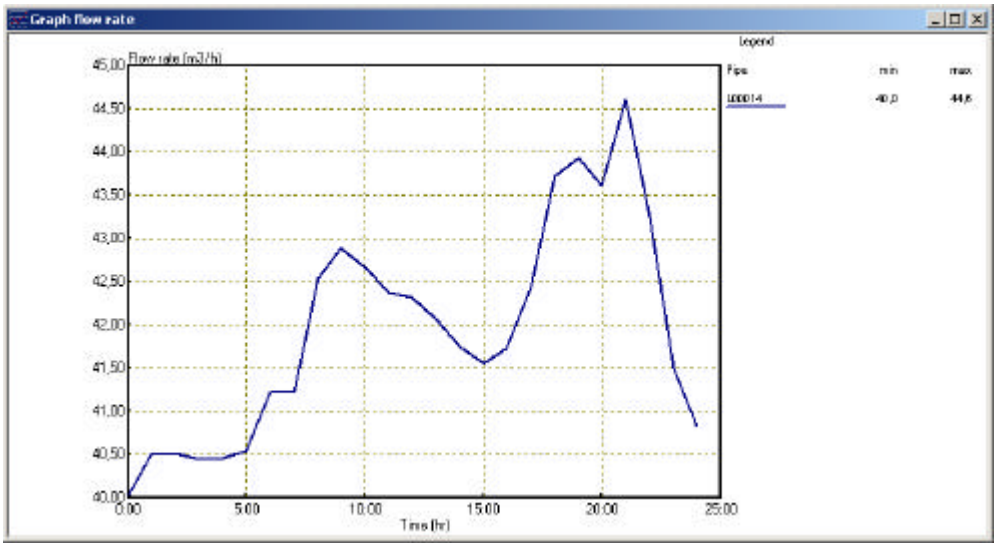


Figure 66 Time series for pipe: flow rate graph

5.9.3 Demand

This option is intended as support in network reliability assessments (pressure related demand calculations).

A diagram (with legend) is displayed, showing entered (initial) and calculated (possibly reduced) demand for a selected node during a certain period (see also Figure 64). Moreover, the percentages of demand not delivered will be calculated for the whole (selected) interval. The node selection can be done on the map by mouse, as well as from the list of the nodes.

5.9.4 Pump curve

This command shows the diagram for a selected pump unit. When the input information has been retrieved, the basic pump characteristics (Q-H) will be displayed (pump speed = 1). The curve will be readjusted with the current pump speed if the output information is in use. The pump unit selection can be done on the map by mouse (clicking at pump symbol), or from the list.

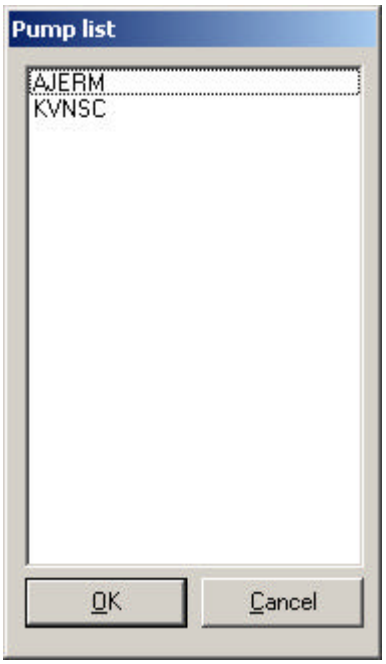


Figure 67 Pump list, from which pump curve can be selected

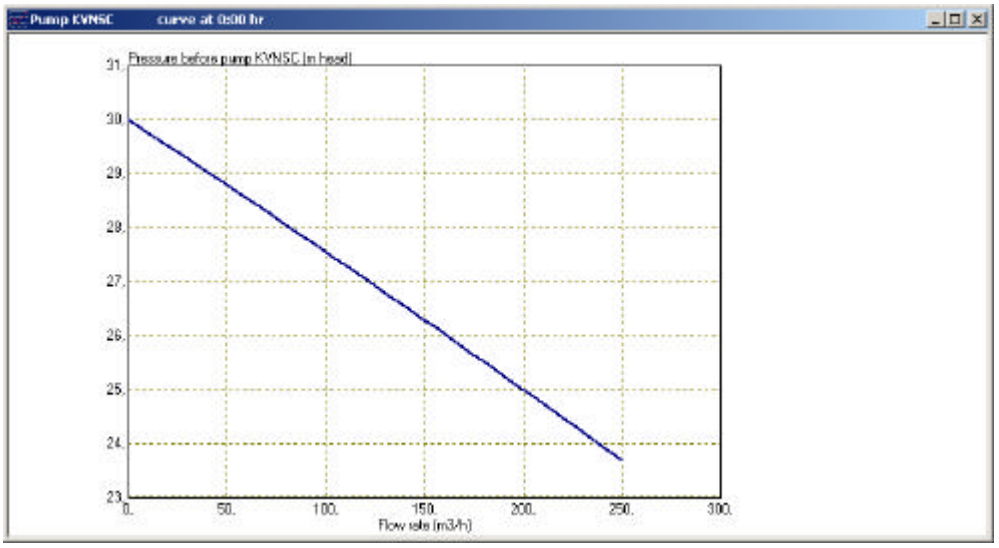


Figure 68 Resulting pump curve

5.9.5 Options

This option adds the data obtained by the measurements in the system to the time series diagrams. The data will be retrieved from the .DAT-file (for the lay-out of the file see section 4.15). The scale of the graph can be adjusted, manually if wished, then the "Auto scale" option should be cancelled. The **Graph**, **Options** command is accessible only from a graph window.

5.10 Main menu: option “Report”

5.10.1 Any file

This command gets a selected file in the viewer. The content can be inspected (but not modified) by clicking the arrow buttons or dragging the mouse over the track-block.

5.10.2 Summary output

This option gives the basic information about the latest calculation executed, and possible STATUS info. The content of the $ file will be displayed, with the names of the files involved in the calculation, description of the network configuration, run parameters, calculation accuracy, etc. The $ file is overwritten with every new calculation. To keep consecutive status messages, the file content should be saved under another name.

5.10.3 Snapshot

In this option a table with calculation results for the entire network (all nodes or links) at a specified moment (currently shown in the map) will be displayed. The network parameters are given in columns while the nodes and links are given in table rows.

Nodes at time 6:00 hr

Name	Demand Cat. 1 m3/h	Demand Cat. 2 m3/h	Demand Cat. 3 m3/h	Demand Cat. 4 m3/h	Demand Cat. 5 m3/h	Demand Entered m3/h	Calculated Cons. m3/h	% Supplied
EMMEN	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
VBOS	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
EMCOM	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
KVNSC	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
P$000	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
74	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
KVNKOM	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
ZWIND	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
DALEN	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
AJERM	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
P$001	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
83	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
KRDHRS	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
4EM	16,00	80,60	0,00	0,00	0,00	96,60	96,60	100,00
77ND	0,00	0,00	0,00	0,00	0,00	0,00	0,00	100,00
29KV	42,13	123,77	3,15	0,00	0,00	169,05	169,05	100,00
31WE	7,34	2,02	7,97	0,00	0,00	17,34	17,34	100,00
32NS	4,62	1,98	0,83	0,00	0,00	7,44	7,44	100,00
24EM	2,00	32,25	1,40	0,00	0,00	35,64	35,64	100,00

Figure 71 Presentation of all node data at time = 0:00 hours

The time can be set directly with the slider at the top of the screen, or by double-clicking on the time window next to it.

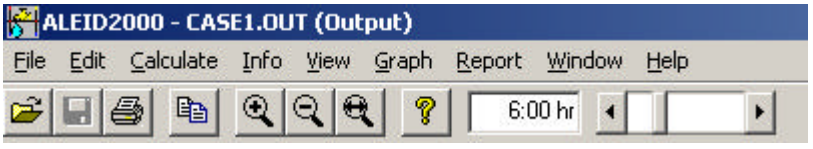


Figure 72 Time slider and hour window at the top of the screen

The information is placed in the table assuming a fixed column width. Depending on the order of values, in some cases it may appear that the values are not properly presented (e.g. the negative sign is missing). To avoid this, the column width should be adjusted where necessary. This can be done by placing the mouse cursor over the line that separates two columns. When the cursor becomes a double arrow, the column width can be adjusted by dragging the cursor (while keeping the left mouse button depressed).

In the case of specific elements, such as reservoirs or pumps, some of the parameters shown are meaningless, so those columns will remain empty. In the case of pumps, the pump power is calculated and presented, instead of the flow velocity.

A selection can be made in the tables. For these details refer to the menu options **Report**, **Filter** and **Report**, **Default**.

5.10.4 Time Series

In this option a table with information for a selected node or link during a specified interval of simulation will be shown. The table manipulation is done in the same way as described for the **Report**, **Snapshot** option.

39VE table

Time hr	Demand Cat. 1 m3/h	Demand Cat. 2 m3/h	Demand Cat. 3 m3/h	Demand Cat. 4 m3/h	Demand Cat. 5 m3/h	Demand Entered m3/h	Calculated Cons. m3/h	% Su
0:00	21,74	2,20	9,06	0,00	0,00	33,00	33,00	100,(
1:00	13,72	1,19	8,03	0,00	0,00	22,94	22,94	100,(
2:00	13,91	1,23	7,83	0,00	0,00	22,97	22,97	100,(
3:00	10,94	1,11	7,72	0,00	0,00	19,77	19,77	100,(
4:00	6,34	1,45	11,48	0,00	0,00	19,27	19,27	100,(
5:00	8,61	1,72	12,02	0,00	0,00	22,35	22,35	100,(
6:00	27,05	3,18	14,82	0,00	0,00	45,05	45,05	100,(
7:00	65,23	4,46	24,00	0,00	0,00	93,69	93,69	100,(
8:00	96,10	6,03	26,04	0,00	0,00	128,17	128,17	100,(
9:00	101,99	6,51	26,24	0,00	0,00	134,73	134,73	100,(
10:00	98,17	6,44	24,16	0,00	0,00	128,77	128,77	100,(
11:00	90,66	6,66	20,43	0,00	0,00	117,75	117,75	100,(
12:00	96,62	5,99	17,10	0,00	0,00	119,71	119,71	100,(
13:00	87,23	6,61	13,31	0,00	0,00	107,16	107,16	100,(
14:00	71,70	6,89	17,51	0,00	0,00	96,10	96,10	100,(
15:00	61,54	6,96	20,85	0,00	0,00	89,36	89,36	100,(
16:00	69,70	7,53	23,78	0,00	0,00	101,00	101,00	100,(
17:00	88,98	7,31	28,36	0,00	0,00	124,65	124,65	100,(
18:00	112,02	7,49	38,52	0,00	0,00	158,03	158,03	100,(

Figure 73 Time series for selected node

5.10.5 Filter

This button allows selective presentation of the tables based on the search criteria defined.

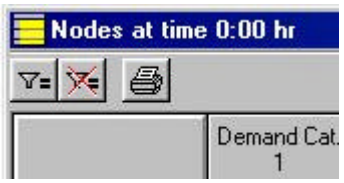


Figure 74 'Filter', 'Show All' and 'Print'buttons

5.10.6 Default

This option restores the standard table format (settings).

5.11 Main menu: option "Window"

This option gives an overview of the open windows. The list with window names will be displayed, so that the selected window can be brought in front of the others.

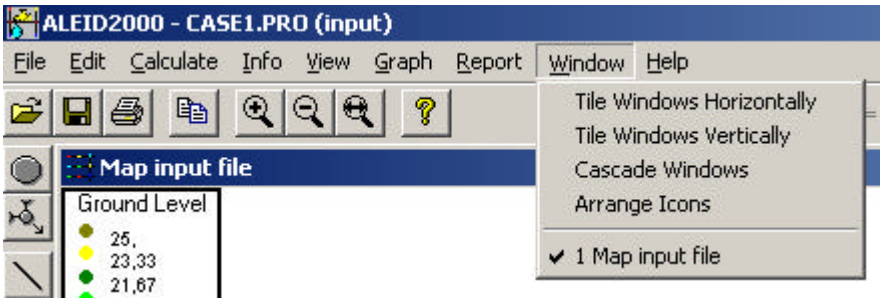


Figure 75 Window menu

5.12 Main menu: option "Help"

5.12.1 About ALEID2000

This option opens a window with general information about the current ALEID2000 version, registration number, Windows version and memory use.

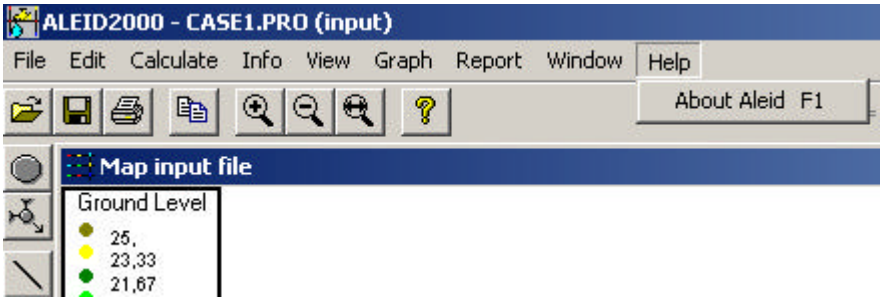


Figure 76 Selecting 'About Aleid2000'

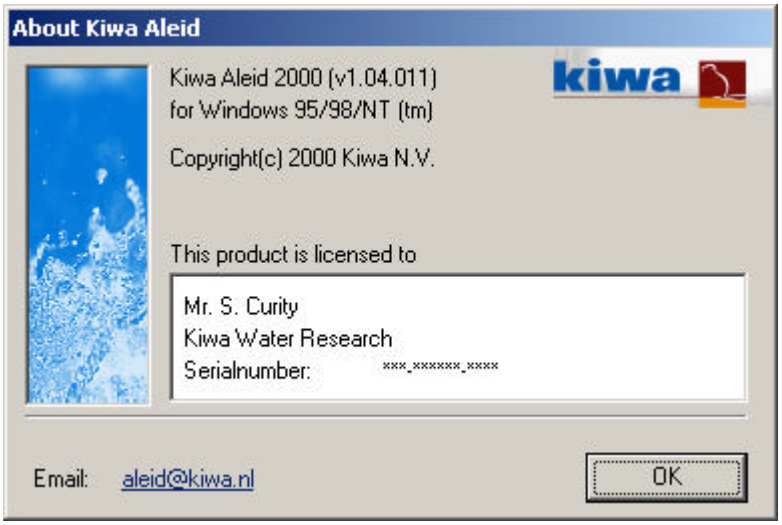


Figure 77 'About Aleid2000' screen

6 Example

6.1 Introduction

A simple example is presented in this Chapter, to demonstrate some important features of ALEID2000, namely:

- creation of original input files;
- execution of the hydraulic calculation;
- input data modification using ALEID2000 menus;
- use of the editor;
- pressure related demand calculation;
- water quality calculation (calculation of travel times);
- presentation of results.

Step 1. Double click on the icon of the ALEID2000-programme group.

Step 2. Double click on the icon of the ALEID2000-programme.

A project file should be opened in the ALEID2000-shell.

Step 3. Click **File**.

Step 4. Click **Open Project**.

The window **Open project file** appears.

Step 5. Select the sub-directory with the ALEID2000 files.

All files present in the sub-directory appear in the **File name** box (extension .PRO).

Step 6. Double click “CASE-96.PRO”.

The window **Open project** appears. In addition to the name of the project file, CASE-96.PRO, other files related to the model will be listed here. These are:

CASE-96.BAS (basic file)
CASE-96.PAT (demand pattern library)
CASE-96.STF (control settings file)
CASE-96.XY (co-ordinates file)
CASE-96.PHF (curves library file)

Step 7. Click “OK”.

The input files are retrieved now and the program proceeds with the map window (**Map input file**). The graphic presentation of the network will be given here, so that model integrity can be checked. The calculation can be performed next.

Step 8. Click **Calculate**

Step 9. Click **Standard**

The dialog box for the selection of run options appears (**Calculation period**). The run time field has to be completed here. The option **Time** gives the possibility to carry out a snapshot calculation for a selected moment. In this case however, a 24-hour simulation is going to be carried out.

Step 10. Click **From** .. **through** ..

Step 11. Click on the input cell to the right of **From** and enter the value 0.

Step 12. Click at the input cell on the right of **through** and enter the value 24.

Step 13. Enter the calculation year in the same way.

Step 14. Enter the calculation time step (default setting 1 hour).

The time units used for the input in **From** .. **through** .., **Time** and **Time step** can be modified by clicking the arrow buttons on the right hand sides. All times in this example are to be set in hours.

The iteration process during calculation is controlled by the input in the **Accuracy** and **Max. number of iterations** cells. The accuracy in this example should be set to 0.01 for flows (relative accuracy), and to 0.001 mwc in case of heads. The maximum number of iterations is 40, in case the requested accuracy cannot be achieved.

Step 15. Fill in the **Accuracy** and **Max. number of iterations** cells.

The names of the output files should be indicated in the fields **Hydraulic solution file name** and **Binary report file name**, together with the report options. As the file extensions are fixed, only the (arbitrary) file name should be typed in the cell. In this example this is “CASE-96” in both cases.

Step 16. When the input is completed, click “OK”.

The calculation will now be executed. A DOS window is opened and the ALCALC program runs.

If the calculation is completed successfully, the status report (the **Output Summary** window) will appear, together with the results for the initial time displayed on the map (**Chart for 0:00 hr**) with the **Colour Coding** setting, possibly from the last session). The results to be shown can be chosen by the menu option **View**, **Colour Coding.** The most common parameter is nodal pressure.

Step 17. Choose the following options from the main menu: **View**, **Colour Coding** and select **Pressure** from the list of node parameters.

Step 18. Select **Adjust to node parameter** from the list of link parameters.

The colour along the pipes will be adjusted accordingly, which gives a better impression of the pressure distribution in the network.

Step 19. Click "OK".

The colour presentation can be adjusted by the main menu option **View**, **Change Legend**.

To inspect the snapshot results at any other moment, the small time scale at the top of the screen can be used. As with files selection, a preferred time can be selected by double-clicking on the hours window or by dragging the slider to the left or right. The exact time (position of the box) is automatically registered and the results for that moment will be shown as soon the left mouse button is released.

The relation between the demand and pressure is defined per node, in theory. In this case, a global pressure/demand relation is going to be introduced. As an illustration, the input file modification will take place via the editor. It is assumed that this editor is already defined by using the Windows options (see sections 3.3.2 and 5.5.1).

Step 20. Chose option **Edit**, **Editor** from the main menu.

Step 21. Double click "CASE-96.STF" (This is the control settings file where the pressure related demand has to be defined, amongst others.)

The editor runs and loads the indicated file. The content has to be modified by adding the [PRESS. DEPENDENCE] section, after the already present section [CONTROLS].

Step 22. Extend the file CASE-96.STF with the following rows:

```
[PRESS. DEPENDENCE]
GLOBAL 20
```

This enables a global (linear) pressure/demand dependence below the nodal pressure of 20 m head (= pressure threshold).

See paragraphe 7.13 and appendices E3.4 and E7 for details.

Step 23. Exit the editor by saving the modified CASE-96.STF file.

The program returns to the output mode. The new calculation can be performed by repeating steps 3 to 16.

NOTE: The above modification can also be done via the menu option **Edit** - **Edit Input** instead of the editor (steps 20 to 23). To do this, the program should switch to input mode first (steps 3 to 7). After this is done, the **Hydraulic Options** option should be selected from the menu, after **Edit**, **Edit Input**. The global pressure threshold can then be set in the dialog box that follows. Possible deviations from this value in specific nodes are specified individually. Before a new calculation is executed (steps 8 to 16), the input modification should be saved. This is done with the menu option: **File**, **Save Input**.

Step 24. Perform the new calculation (steps 3 to 16).

The calculation results show positive pressures in the system. This is achieved by the (calculated) demand reduction. Menu option **Graph**, **Time Series Node** offers a good impression of the percentage of demand delivered.

Step 25. Set the time at 0:00 hours (on the time scale at the top of the screen).

Step 26. Choose options: **Graph**, **Time Series Node** from the main menu. Select the **% of demand supplied** from the list. Press "OK".

Step 27. Perform the selection by clicking on a maximum of 5 nodes (include node "A" in this example!) and exit the selection mode by pressing the right mouse button.

The graph appears. Node "A" shows a particularly high demand reduction at a given moment. To gain more insight, a graph can be made showing the nominal (initial) demand together with the calculated (reduced) demand during the entire simulation period. To select a node, choose the option **Graph**, **Demand** from the main menu.

Finally, a simple water quality calculation is going to be carried out, as an illustration of the program features. To be able to do that, a water quality file (extension .WQI) should be created. The editor can be used here, as well.

Step 28. Chose **Edit**, **Editor** from the main menu.

Step 29. Type the file name of the new water quality file in the cell: CASE-96.WQI.

Step 30. Click “OK”.

The editor runs. The file input is going to be modified for a calculation of travel times in the network.

Step 31. Type the following rows:

```
[QOPTIONS]
AGE
```

Step 32. Exit the editor after saving the file content.

Step 33. Run a new calculation (steps 6 to 19), after adding the CASE-96.WQI in the project file (**Open Project** dialog box, “Water qual. file” cell)

NOTE1. The file is not automatically retrievable, once added to the list in the project file. To allow this, the “Save and Retrieve” option from the **Open Project** dialog box has to be chosen.

NOTE2. The fields in the **Water quality** tab of the **Calculation** dialog box (menu option **Calculate**, **Standard**) should be completed, in principle. If a water quality calculation has already been done, the values from the last calculation will appear. For calculations using default settings, the number of segments will be displayed only (standard = 100). The calculation is then performed in compliance with the principle explained in Appendix E section E.6.8, concerning time steps and residence times.

NOTE3. The outputs of hydraulic calculations performed earlier can also be used for the water quality simulation. Option **None** has to be selected in the run time field of the **Calculation** dialog box (menu option **Calculate**, **Standard**). The name CASE-96 has to be indicated as the previously created .HBI-file. A file name other than CASE-96 (.OUT) should be used for the binary report in order to prevent the CASE96.OUT file from being overwritten.

Step 34. Inspect the travel times after the calculation has been successfully completed, e.g. with the menu option **Graph**.

7 Reference

7.1 [CONSUMPTION]

Purpose:
Information about demand nodes and their average demand(s). Connecting nodes (without demand) should also be included here.

Format:
```
node  number
   qavg  code1 code2 code3 code4 code5
```

Parameters:

node	:	node name, up to 7 alphanumeric characters (incl. space)	
number	:	number of demand categories (maximum 5 per node), this determines the number of rows which follow, each with the same format:	
qavg	:	average demand	
code1	:	identification number of the daily demand pattern	00-99
code2	:	identification number of the weekly demand pattern	00-99
code3	:	identification number of the leakage percentage	00-99
code4	:	identification number of the correction factor	00-99
code5	:	identification number of the growth pattern	00-99

Remarks:
- The average demand can be specified in m^3/hour or in American units, see Appendix H.
- The identification numbers refer to the various patterns given in the demand pattern library. Selected patterns adjust the average demand to the nominal demand for a given node at a given moment.
- Identification 00 means that the default setting will be used for adjustment of the average demand. Thus, there can be a maximum of 99 different demand patterns (categories) specified in the program.
- The number of demand categories for a connecting node equals 0.
- After modifications, the shell will save the comment on the last row only.

Example:

```
[CONSUMPTION]
Node   2      ; two demand categories
30     1      2      0      0      0
10     2      1      1      1      1
```

7.2 [CONTROLS]

Purpose:
Description of time- or status-dependent changes in the calculated network. Most commonly this is related to the operation of pumps and valves.

Format:

```
LINK  pipe  setting  AT  TIME  tvalue  (units)
LINK  pipe  setting  IF  NODE  cnode   BELOW   clevel
LINK  pipe  setting  IF  NODE  cnode   ABOVE   clevel
LINK  pipe  setting  IF  LINK  cpipe   BELOW   cflow
LINK  pipe  setting  IF  LINK  cpipe   ABOVE   cflow
TANK  node  setting  AT  TIME  tvalue  (units)
```

Parameters:

pipe : pipe name
node : node name of a fixed head node or a reservoir
setting : modified setting. This can be:
- a pump status (OPEN or CLOSED);
- a pump speed (relative, related to the speed used for the definition of the pump characteristics);
- a valve setting (pressure, flow or head-loss, or OPEN or CLOSED);
- a pipe / shutter status: OPEN or CLOSED
- a value of the fixed head (in mwc) by the TANK-control.

tvalue : time (moment) in which control changes
units : optional units for the selected time step:
SEC : seconds
MIN : minutes
HOURS : (default setting)
DAY : days
cnode : name of the control node
cpipe : name of the control pipe
clevel : critical level at which control changes (head, in mwc)
cflow : critical flow rate for which control changes (in flow units)

Remarks:
- One control setting occupies one row only (in the file).
- More control settings can be applied to the same pipe.
- The control can be set on time or the water level in a reservoir, on the pressure in a node, or on the flow in a pipe.
- times can be specified as *hours:minutes* or with decimal point. In the second case, the units assumed are hours, unless specified differently.
- No optimization of the specified controls is possible by the program. For calculation of a new time, the program will survey the control settings and apply those relevant for that moment. Head and flow related controls will be handled sequentially. Thus, the “and/and” and “and/or” constructions for combinations of head- or flow controls are not possible.
- Head and flow controls take priority over time controls.

- Separate on and off controls have to be given by the head and flow controls, e.g. when a pump has to switch on if a head somewhere rises above 30 mwc (control defined as: LINK *pump* OPEN IF NODE *cnode* ABOVE 30) it will not switch off once the head drops below 30 mwc. This has to be specified separately.
- Head and flow controls will be effectively applied only if the accuracy specified for the calculation (iteration) can be achieved.
- Unlike the demand patterns, the control settings are not periodic. For each simulation it should be checked whether all relevant settings are defined for the entire period.

Example:

```
LINK  pipe1  CLOSED   AT  TIME  3
LINK  pump2  OPEN     IF  NODE  node3   ABOVE   35
LINK  pump2  CLOSED   IF  NODE  node3   BELOW   40
TANK  resv1  35       AT  TIME  6       ; fixed
                                        head of
```

7.3 [COORDINATES]

Purpose:
Defines X and Y co-ordinates and node ground elevations for all nodes listed in basic configuration.

Format:
```
name  x  y  z
```

Parameters:

name	:	node name
x, y	:	x and y co-ordinate for the purpose of graphical presentation
z	:	ground elevation for the determination of nodal pressures from the calculated heads (in meters above reference level)

Remarks:
- Arbitrary coordinate system and units can be adopted for the X and Y co-ordinates.

Example:

```
[COORDINATES]
node1    2315.6  6000.2   5
node2    4564.3  2353.2   3
```

7.4 [CORRECTION]

Purpose:
Information about the correction factor. This factor is applied to the average demand.

Format:
```
nr.   multiplier
```

Parameters:

nr.	:	pattern identification number (01-99)
multiplier	:	correction factor

Remarks:
- Average demand which follows pattern number 00 will remain constant during simulation (default setting, correction factor = 1.0).

Example:
```
[CORECTION]
09       1.02    ;2% demand increase
```

7.5 [DAYFACTORS]

Purpose:
Information about the day factors. These factors are applied to the average demand.

Format:
```
nr.   multipliers
```

Parameters:
nr. : pattern identification number (01-99)
multipliers : factors (for a maximum of 7 days)

Remarks:
- Average demand which follows demand pattern number 00 will remain constant during simulation (default setting, peak factor = 1.0).
- If one pattern is shorter than the specified simulation period, it will be repeated from the beginning, with the factors for day 1, 2, etc.

Example:

```
[DAYFACTORS]
03       1.2     1.3     ;pattern for two
                         days
```

7.6 [END]

Purpose:
Conclusion of the basic file

Format:

[END]

Remarks:
- The section [END] contains the header only. All data or comments following the [END] header will be neglected.

7.7 [GENERAL]

Purpose:
Information about the simulation run parameters that do not result directly from the model input.

Format:

START	begtime (units)
STOP	endtime (units)
YEAR	year
TIMESTEP	timestep (units)
TRIALS	value
ACCURACY	value1 value2

Parameters:

begtime	:	start time
endtime	:	end time
year	:	specific year of supply
timestep	:	time interval between two consecutive calculations
units	:	time units, default setting = HOURS
value(1,2)	:	numerical value

Remarks:

- START
  Start of the calculation at specified time (moment)
- STOP
  End of the calculation at specified time (moment)
  If the values for START and STOP are equal, one steady state (snapshot) calculation will be executed.
- YEAR
  The year for which the modelled supply conditions are valid.
- TIMESTEP
  Time (calculation) interval between two consecutive steady state calculations
- TRIALS
  Maximum allowed number of iterations if the requested accuracy cannot be reached. The default setting is 40.
- ACCURACY
  Specifies calculation accuracy. A double criterion is applied. The criterion defined with “value1” ends the calculation when the relative error of the flow rates becomes smaller than the specified value, for all links. Expressed numerically, with $Q_k(i)$ being the flow rate in pipe "i" after iteration "k":

$$\frac{|Q_{k+1}(i) - Q_k(i)|}{|Q_{k+1}(i)|} < \text{value1} \qquad \textbf{(1)}$$

  The default setting for this criterion is 0.01. Values smaller than 10^{-5} cannot be used.

The criterion specified with “value2” ends the calculation, if the head difference between two consecutive iterations becomes smaller than the given value, for all nodes. Expressed numerically, with $H_k(j)$ being the head in node "j" after iteration "k":

$$|H_{k+1}(j) - H_k(j)| < \text{value}\,2 \qquad \textbf{(2)}$$

The default setting for this criterion is 0.001 mwc.

Both criteria have to be satisfied before the iteration will stop.

Example:

```
[GENERAL]
START     0
STOP      12
YEAR      2000
ACCURACY  0.02    0.001
```

7.8 [HYDRANTS]

Purpose:
Information about hydrants in the model.

Format:
```
hyd   net   out    hydID
```

Parameters:

hyd	:	hydrant name
net	:	name of the network node where the hydrant is connected
out	:	name of the outflow node; this has to be a fixed head node (from the [TANKS] section)
hydID	:	identification number of a hydrant characteristics from the curves library, attached to the hydrant (Q, H-relation)

Remarks:
- The hydrant regime has to be specified in the [CONTROLS] section of the control settings file.
- Atmospheric pressure should be used as the output pressure. Elevation of the hydrant output (fixed head node) should be taken into account. Losses at the stand pipe etc. have to be included in the Q, H-relation of the hydrant.

Example:
```
[HYDRANT]
HYDRANT   Node1   Node2   6 ; Hydrant nr. 6
```

7.9 [LEAKAGE]

Purpose:
Information about the leakage percentages. The percentage specified is recalculated by the program into the multiplier of the average demand. E.g. for a leakage of 7 % of the total delivery, the factor becomes 1.075, viz.
100/(100-7).

Format:
```
nr.   %
```

Parameters:
nr. : pattern identification number (01-99)
% : leakage as a percentage of total delivery

Remarks:
- Average demand which follows leakage pattern number 00 will remain constant during simulation (default setting, leakage factor = 1.0).
- The multiplier calculated from the leakage percentage is applied to the average demand influenced by other (peak) factors as well. Example: for an average demand = 3 m^3/h, and leakage of 50% (factor = 2):

Case 1:	**Case 2:**
if peak factor = 0.5	if peak factor = 1.5
total demand: 3*2*0.5 = 3 m^3/h	total demand: 3*2*1.5 = 9 m^3/h
where: 1.5 m^3/h consumption	where: 4.5 m^3/h consumption
1.5 m^3/h leakage	4.5 m^3/h leakage

Thus, despite the constant percentage, in this way the leakage becomes demand pattern dependent i.e. expressed in m^3/h, lower during the minimum hour supply than the maximum hour supply. This conflicts with reality where the leakage (flow) is predominantly pressure related, i.e. higher at night. Hence, in networks with high leakage percentages, this modelling approach should be applied with caution. An alternative for such cases is to model leakage as a separate category with its own demand pattern.

Example:

```
[LEAKAGE]
01        3       ; 3% leakage
02        10
```

7.10 [OPTIONS]

Purpose:
Information on other settings required for hydraulic calculation.

Format:
```
UNITS              option
HEADLOSS           option
SPECIFIC GRAVITY   value
VISCOSITY          value
```

Parameters:
option : a choice from a fixed list of options
value : a numerical value

Remarks:
- The input in this section is required only if the values deviate from the default settings.
- UNITS specifies the units in which flow rates and pressures (heads) will be displayed (see Appendix H).
  The options are:
  SI : m^3/hour for flow, mwc for pressure (standard setting);
  GPM : gallons per minute, feet (of water column);
  CFS : cubic feet per second, feet;
  MGD : million gallons per day, feet.
  Care should be taken to chose the appropriate set of units, which corresponds to the data from the input files.
- HEADLOSS specifies which head-loss formula will be used in the calculations.
  The available choices are:
  D-W1 : Darcy-Weisbach iterative (standard)
  D-W2 : Darcy-Weisbach approaching
  H-W : Hazen-Williams
  C-M : Chézy-Manning
  The theoretical concepts behind these approaches are discussed in Appendix F section F.4.2.
  It should be decided in advance which coefficient to use for the wall roughness description in the input. The standard parameter is the k-value.
- SPECIFIC GRAVITY is the weight per unit volume of the fluid being modelled, relative to that of water. The default setting value is 1.0.
- VISCOSITY is the kinematic viscosity of the water. The units for this parameter are m^2/sec (or ft^2/sec). The standard value is $1.31*10^{-6}$ m^2/sec, corresponding to a water temperature of 10^oC. Viscosity is used only in head loss calculations by the Darcy-Weisbach formula or when the pipe wall reaction mechanism is used for water quality calculations.
- This section can appear in the file only once.

Example

```
[OPTIONS]
HEADLOSS  D-W2
UNITS     SI
```

7.11 [PATTERNS]

Purpose:
Information about pattern factors.

Format:
```
nr.   multipliers
```

Parameters:

nr.	:	pattern identification number (01-99)
multipliers	:	factors that multiply corresponding average demands

Remarks:

- Average demand which follows demand pattern number 00 will remain constant during simulation (default setting, peak factor = 1.0).
- A row always begins with the pattern number, followed by the factors. Factors from the same pattern can occupy several rows, if at the beginning of each, the same pattern number is specified. The description of the factors for a new pattern begins with the new pattern number.
- The number of factors in a row is limited by the maximum number of characters allowed, which is 200.
- The length of the pattern is variable, for different patterns different lengths can be specified.
- If one pattern is shorter than the specified simulation period, it will be repeated from the beginning. E.g. for a simulation duration of 25 hours and a pattern from 0-23 hours, the program proceeds with the calculation of hour 24 by taking value for hour 0 and for hour 25 by taking the value of hour 1.
- Simulation runs are in principle done starting from 0 (hours). If a later time is chosen to begin from (say 6 hours), the program selects corresponding peak factor from available pattern(s), as a start value.
- The comment after the end row of a pattern will be saved by the program.

Example:

```
[PATTERNS]
01  1.0   1.2   1.4  1.5   1.5   1.23
01  1.34  1.67  1.8  1.69  1.54  1.2 ; 12 values
03  1.0   1.0   1.0  1.1   1.1   1.1
03  1.5   1.5   1.5  1.0   1.0   1.0
```

7.12 [PIPES]

Purpose:
Information about ordinary pipes.

Format:
```
pipe  from  to   diam  length  rcoeff (llcoeff)
(CV)
```

Parameters:

pipe	:	pipe name, max. 7 alphanumeric characters
from	:	begin node name
to	:	end node name
diam	:	internal diameter (mm)
length	:	length (m)
rcoeff	:	wall roughness factor (e.g. k-value (mm) for Darcy-Weisbach head-loss formula)
llcoeff	:	local loss coefficient (-)
CV	:	indication of the presence of a check valve on the pipe

Remarks:
- Pipe name is compulsory.
- "rcoeff" depends on the type of calculation. The default setting is an iterative form of the Darcy-Weisbach equation. This setting can be changed via the [OPTIONS] section of the control settings file.
- The order of node names is important if a check (non-return) valve is installed: the begin node is the upstream and the end node the downstream node.
- Otherwise, when the flow direction in the pipe is reversed (i.e. from the end node to the begin node), the tabulated results for flow rates and velocities will be presented as negative values.

Example:

```
[PIPES]
Testpipe  BeginNo  EndNo  300  500  0.05 ;

Pipe67    FromNo   ToNo   100  5    0.1  1.2 ; with
local loss

Strl20    Hou1     Hou2   200  5    0.1  CV ; check
valve present

Strl20    Hou1     Hou2   200  5    0.1  1.5  CV ;
check valve & local loss
```

7.13 [PRESS.DEPENDENCE]

Purpose:
Gives the list of nodes where the pressure related demand should be introduced, with the corresponding value of the pressure threshold. The theoretical concepts behind the pressure related demand being applied in this program are discussed in Appendix F section F.3.4.

Format:
```
GLOBAL setting
node   setting
```

Parameters:
setting : pressure threshold (mwc)
node : name of the node in which the pressure related demand is applied

Remarks:
- The threshold specified after GLOBAL will be applied in the whole network, with the exception of the individual nodes mentioned further.
- The GLOBAL setting is also valid for "old" type 50-nodes from ALEID 7.1(L) when these files are transferred to ALEID2000. Unlike the ALEID2000 concept, pressure dependent calculation in ALEID 7.1(L) would not affect the nodes with constant demand (labelled there as type 50). If this is to be maintained in ALEID2000 as well, those nodes should be included in the list below the GLOBAL setting, with the threshold values set to 0.
- Only one node with a corresponding pressure threshold can be specified in each row.
- The node name must exist in the basic file.

Example:

```
[PRESS.DEPENDENCE]
GLOBAL  20 ; general threshold of 20 mwc
CWR1    10 ; exception: clear water reservoir
```

7.14 [PUMP CURVES] and [HYDRANT CURVES]

Purpose:
Description of Q, H-relation for each pump unit or hydrant in the system.

Format:

```
nr.   Type  #rows
Qmin  Qmax    a  b  c
```

or

```
nr.   Type  #rows
Qi Hi
```

Parameters:

nr.	:	Pump/hydrant curve identification number
Type	:	type of the specified Q, H-relation: QH definition by means of one or more quadratic curves (definition by formula) P curve definition by a number of Q, H-points with linear interpolation in between (definition via polygonal line)
#rows	:	number of the following rows
Qmin, Qmax	:	specifies the flow range within the Q, H-relation is valid (for Type = QH) (m^3/h)
a,b,c	:	coefficients of the formula $H = aQ^2+bQ+c$ (for Type = QH)
Qi,Hi	:	Q, H-set of values for point "i" of the Q, H-curve (Q in m^3/h, H in mwc).

Remarks:
- A maximum of 15 rows can be occupied for the description of one curve (max. 15 Q, H-points).
- One row should contain only one a, b, c or one Q,H set of data.
- The list of Q, H-points must show ascending values for flow rates.

Example:

```
[PUMP CURVES]
1  P  6
   0     104
   300   103.58
   600   102.58
   900   101.08
   1200  99.15
   1500  96.80
2  QH 1
   0      3100    -0.0000370    0.0    333.335
```

```
[HYDRANT CURVES]
1  QH 2
   0  20    0.0      0.04891 1.00
   20 200   .003186 -.087669   2.45
```

7.15 [PUMPS]

Purpose:
Information about pumps and boosters.

Format:
```
pump  suct  press  pumpID
```

Parameters:

pump	:	pump name
suct	:	name of the suction node
press	:	name of the pressure node
pumpID	:	identification number of a pump characteristics from the curves library, attached to the pump (Q, H-relation)

Remarks:
- The pump regime has to be specified in the [CONTROLS] section of the control settings file.
- A booster out of operation will be modelled as a closed pipe.

Example:

```
[PUMPS]
Pump1  FromSuc  ToPress  5 ; pump curve nr. 5
```

7.16 [QOPTIONS]

Purpose:
Defines the type of water quality calculation intended for modelling.

Format:
```
NONE or
CHEMICAL or
substance   (units)
AGE or
TRACE node
```

Parameters:
- substance : arbitrary substance (name) given by the user
- node : name of the node from which the water route is followed
- units : alternative units for the water quality calculations. The default setting is mg/l

Remarks:
- Model simulation can be executed with or without one selected water quality calculation:
  NONE - no water quality calculation included in the simulation
  CHEMICAL - concentration calculation of a chemical substance
  substance - ditto, declaring a particular substance
  AGE - calculation of average travel time
  TRACE - calculation of total percentage of the specified source (node) for the entire period of simulation
- Hence, the alternative for the word CHEMICAL can also be the name of the substance concerned. This name will appear in the output of the calculation.
- These sections can be specified only once in the file.
- The calculation of the travel time keeps the travel time in a feeding point at zero. If so desired, an initial value for the travel time can be specified under the header [QUALITY].

Example:

```
[QOPTIONS]
TRACE source1 ; water path source1 will be followed
```

7.17 [QPATTERNS]

Purpose:
Defines patterns of the concentration variation.

Format:
```
nr.   multiplier
```

Parameters:
nr. : pattern identification number
multiplier : multiplication factor

Remarks:
- The same time step used for the demand patterns has to be applied for these patterns.
- The length of the patterns is arbitrary. Different patterns may have different lengths.
- A maximum of 99 patterns can be specified in the file.

Example:

```
[QPATTERNS]
01  0.98    1.02    1.05    1.4
```

7.18 [QTIMES]

Purpose:
Defines a number of additional parameters for making a water quality calculation.

Format:

QUALITY TIMESTEP	time
MINIMUM TRAVELTIME	time (units)
SEGMENTS	value

Parameters:

value	:	numerical value
time	:	duration

Remarks:

- QUALITY TIMESTEP
  Specifies the time interval for the calculation of the water quality parameters. If this is not given, the program will use an internally calculated time interval based on the shortest travel time occurring in the pipes. The default setting = HOURS.
- MINIMUM TRAVELTIME
  Specifies the shortest travel time detectable by the program. Shortest travel times will be set to the specified value. Travel times through pumps and valves are considered zero and are not influenced by this limit. The default setting for the minimum travel time is set to 1/10 of the time interval specified for hydraulic calculations.
- SEGMENTS
  Specifies the maximum number of segments in which a pipe can be divided during the water quality calculations. The standard value is 100. The number of pipes that reach this limit can be requested in a status report by [REPORT] section in ALCALC.INI (at the beginning of calculation).

Example:

```
[QTIMES]
SEGMENTS 300
```

7.19 [QUALITY]

Purpose:
Defines the initial quality in the network nodes at the beginning of the simulation.

Format:
```
node  quality
```

Parameters:
node : node name
quality : initial quality (concentration in mg/l for chemical substances, hours for travel time, or percentage for source tracing)

Remarks:
- Each row specifies the quality for one node only.
- The unit of water quality is dependent on the type of calculation performed.
- When nothing is specified for a node, a default setting, 0, will be applied.

Example:

```
[QUALITY]
node3 1.00 ; concentration in mg/l
```

7.20 [REACTIONS]

Purpose:
Defines the reaction coefficients used for the water quality reaction (see Appendix F section F.6.8).

Format:

```
DIFFUSITY           value
GLOBAL BULK         bulkcoeff
GLOBAL WALL         wallcoeff
BULK  pipe          bulkcoeff
WALL  pipe          wallcoeff
TANK  node          bulkcoeff
LIMITING POTENTIAL value
```

Parameters:

bulkcoeff	:	bulk reaction coefficient of the reaction, day^{-1}
wallcoeff	:	wall reaction coefficient of the reaction, m/day
pipe	:	pipe name
node	:	node name
value	:	numerical value for the 'limiting potential' in mg/l

Remarks:

- DIFFUSITY gives the molecular diffusion of the chemical substance which is monitored. The diffusion will be expressed in m^2/sec or ft^2/sec depending on the selected units. The standard value is 1.21×10^{-9} m^2/sec, which is chlorine diffusion in water at 20 ^oC. Diffusion will only be used if pipe wall reactions are taken into consideration for the water quality calculation.
- GLOBAL gives a single coefficient for the bulk or wall coefficient, which applies for the whole network, with the exception of individually specified pipes.
- BULK and WALL provide coefficients for individual pipes.
- TANK gives a reaction coefficient for an individual tank. This takes priority over the given GLOBAL BULK coefficient.
- The units of the WALL and the BULK coefficient are different. One way to compare them would be to divide the wall coefficient by the hydraulic stream of the pipe (half of the standard stream). The resulting value will be in the same units as the bulk coefficient.
- The value specified by LIMITING POTENTIAL indicates the maximum concentration possible. When this is being used in combination with a positive reaction coefficient, the growth will become proportional to the difference between the limiting potential and the present concentration. For example, this can be used to simulate the growth of THMs (trihalomethanes). The use of a LIMITING POTENTIAL value with a negative reaction coefficient has no meaning.
- When a decrease in the reaction has to be modelled, a negative number must be used for the coefficients.
- For the modelled reactions see Appendix section F, section F.8.2 and further. A first order reaction is modelled, thus the dependence between the reaction speed and the concentration is linear.

Example:

```
GLOBAL BULK   -.1
GLOBAL WALL   -1.5
BULK  pipe3   -.5
BULK  pipe5   -.5
WALL  pipe3   -.3
TANK  storage1   .05
LIMITING POTENTIAL   10 ; (mg/l)
```

7.21 [RELATIVE GROWTH]

Purpose:
Information related to demand forecasting. The annual growth factors are applied to the original demand. A linear increase model is assumed here. See also the example in Appendix F, section F.3.3.

Format:
```
nr.   year  year  year  year  year
nr.   %     %     %     %     %
```

Parameters:

nr.	:	pattern identification number (01-99)
year	:	each year indicates the beginning of the following prognosis period
%	:	growth percentage for corresponding prognosis period.

Remarks:
- Average demand which follows pattern number 00 will remain constant during simulation (default setting, growth factor = 1.0).
- An equal number of years and growth percentages have to be specified, such that the last percentage specified remains valid for any moment after its corresponding year.

Example:

```
[RELATIVE GROWTH]
02    1990  1995  2000  2010
02    1.0   1.2    0.5    0.0   ; growth percentages
```

Hence, the annual population growths in this example will be: 1.0 % in the period 1990-1994, 1.2 % in the period 1995-1999, 0.5 % in the years 2000-2009 and 0.0 % from year 2010 onwards.

7.22 [REPORT]

Purpose:
Storage of the requested output results in an ASCII-format file.

Format:

STATUS	NO, YES or FULL
BFile	name
PAGESIZE	value
REPORT START	begtime (units)
REPORT TIMESTEP	timestep (units)

Parameters:

name	:	name of the binary file
value	:	numerical value
begtime	:	start time
timestep	:	time interval
units	:	time units, default setting = HOURS

Remarks:

- STATUS NO, YES or FULL
  Specifies the level of information the status report will give about the calculation run. This information will be stored in the ASCII file with name "$". The variables have the following meaning:
  - NO — no extra information will be generated
  - YES — a table is generated for each selected time, containing information about the total number of iterations, inflow/outflow of the reservoirs and on/off status of the pumps. If applicable, a table with information about the water quality calculation will also be generated.
  - FULL — The above information will be extended with the registered accuracy per iteration, for each calculation step. Moreover, the ten nodes with the greatest result deviation will be indicated
- BFile
  Specifies the name of the binary output file used for communication with the Windows shell.
- PAGESIZE
  Specifies the number of rows per page
- REPORT START
  Specifies from which simulation time (moment) the ASCII report and the binary output file should start registering the results.
- REPORT TIMESTEP
  Specifies at which time intervals the results should be registered.

Example:

```
[REPORT]
BFILE   test.out
STATUS FULL
REPORT TIMESTEP 2
```

7.23 [SOLUTION]

Purpose:
Defines the file name(s) where the results of an ordinary hydraulic calculation are kept for (later) use as input into water quality calculations.

Format:
HYDRAULICS USEname
HYDRAULICS SAVE name

Parameters:
name : name of the binary file

Remarks:

- HYDRAULICS USE
  The file name specified by this command contains the information that can be used later when the water quality calculations are requested (without running the hydraulic calculations beforehand).
- HYDRAULICS SAVE
  The binary file specified by this command stores a calculated hydraulic output.
- If the same file name is specified in both cases, it will mean that the latest hydraulic output will be used for water quality calculations. When another is needed, the file names should differ.

Example:

```
[SOLUTION]
HYDRAULICS SAVE test.hyd
```

7.24 [SOURCES]

Purpose:
Information about the basic concentrations at nodes that act as sources in spreading a chemical substance throughout the network.

Format:
```
node  concen  (qpattern)
```

Parameters:

node	:	node name
concen	:	concentration of substance entering the network
qpattern	:	identification number of the concentration variation pattern.

Remarks:
- Each row specifies the quality for one node only.
- This option can only be used for concentrations of a substance.
- When the pattern is not defined, the given concentration will not vary.

Example:

```
[SOURCES]
source1 1.00 3 ; quality pattern nr. 3
```

7.25 [TANKS]

Purpose:
Information about fixed head nodes or reservoirs.

Format:
```
node  ih
   or
node  ih minh  maxh  diam
   or
node  ih tankID
```

Parameters:

node	:	name of the reservoir node, up to 7 alphanumeric characters
ih	:	initial head, i.e. the water level above reference level used to start the calculation (m)
minh	:	minimum head (m)
maxh	:	maximum head (m)
diam	:	tank diameter, for cylindrical tanks (m)
tankID	:	identification number of the volume-head (V, H) relation which describes the reservoir operation

Remarks:
- The level of the fixed head can be changed via the [CONTROLS] section in the control settings file.
- The V, H-relation is defined in the curves library (*.PHF) under the header [TDATA].

Example:

```
[TANKS]
Watert  30   28    34    20 ; cylindrical water
tower
Fhead   40 ;  fixed head node (reservoir)
realres 35   3 ; tank with V,H-curve nr. 3 attached
```

7.26 [TDATA]

Purpose:
Defines the relation between the volume and the water level in the reservoir.

Format:
nr. Vi Hi

Parameters:
nr. : identification number of the V, H-relation
Vi : reservoir volume (m^3), in point "i" of the V, H-curve
Hi : surface water level (m) above reference level, in point "i" of the V, H-curve

Remarks:
- The V, H-data sets create a polygonal line, i.e. linear interpolation will be applied for the values in between the specified ones.
- The list of V, H-points must use ascending values for volumes.

Example:
```
[TDATA]
   2     0        30
   2     6000     40
   2     12000    50
```

7.27 [TIMES]

Purpose:
Defines which time (calculation) step is valid within a 24-hour demand variation and possibly for concentration patterns.

Format:
PATTERN TIMESTEP n (MIN)

Parameters:

n : time step (interval) between two consecutive peak factors. This is usually given in hours; when expressed in minutes the units (min) should be added.

Remarks:
- This section can be given only once in the file.

Example:

```
[TIMES]
PATTERN TIMESTEP   15 MIN  ; interval of 15 minutes
```

7.28 [TITLE]

Purpose:
Description of the specific case (network).

Format:
Up to 9 rows of maximum 80 characters each.

Remarks:
- This section is optional.
- It should appear only once in the file.
- Additional comments after use of ";" do not create problems during data reading, but they will automatically be removed after the input is modified and saved via the ALEID2000 menu (options “Modify” and “File - Save input”).

Example:

```
[TITLES]
This is a test network
This comment will be saved
; this will not be saved by the program
```

7.29 [VALVES]

Purpose:
Information about the valves

Format:
```
valve   from    to diam type    setting (llcoeff)
```

Parameters:

valve	:	valve (pipe) name
from	:	upstream node name
to	:	downstream node name
diam	:	valve diameter (mm)
type	:	valve type: PRV pressure reducing valve PSV pressure sustaining valve PBV pressure breaking valve FCV flow control valve TCV throttle control valve
setting	:	pre-set head (mwc) for PRV, PSV en PBV; pre-set flow for FCV (m^3/h); loss coefficient for TCV (-)
llcoef	:	local loss factor for fully opened valve (equals 0 if not specified).

Remarks:
- The valves are operated via the [CONTROLS] section in the control settings file.
- For description of the valve types see Appendix F, section F.4.4 and further.

Example:

```
Valve Node1 Node2 100 PRV 25
Valve Node1 Node2 100 PRV 25
```

LITERATURE

1. Huisman, L., 1969, *Stromingsweerstanden in leidingen* ('Hydraulic head loss in pipelines', in Dutch), Kiwa publication No. 14, Rijswijk, The Netherlands.
2. Rossman, L.A., Boulos, P.F. and Altman, T., 1993, *The Discrete Volume Element Method for Modeling Water Quality in Pipe Networks*, in: Journal of Water Resources Planning and Management, Vol. 119, no. 5, ASCE.
3. Edwards, D.K., Denny, V.E. and Mills, A.F., 1976, *Transfer Processes*, McGraw Hill, New York.
4. Commissie Leveringszekerheid, 1992, *Aanbevelingen voor de leveringszekerheid van de drinkwatervoorziening*, ('Recommendations on the sustainability of drinking water supply', in Dutch), final report of the Committee on Water Supply Sustainability, VEWIN, Rijswijk, The Netherlands.
5. Rossman, L.A., 1993, *EPANET Users Manual*, EPA-RRL, Cincinnati.