The HYDRUS Software Package for Simulating

the Two- and Three-Dimensional Movement

of Water, Heat, and Multiple Solutes

in Variably-Saturated Media

User Manual

Version 1.0

March 2006

PC-Progress, Prague, Czech Republic

The HYDRUS Software Package for Simulating

the Two- and Three-Dimensional Movement

of Water, Heat, and Multiple Solutes

in Variably-Saturated Media

User Manual

Version 1.0

J. Šimůnek[1], M. Šejna[2] and M. Th. van Genuchten[3]

March 2006

[1]University of California Riverside, Riverside, CA
[2]PC-Progress, Prague, Czech Republic
[3]George E. Brown, Jr. Salinity Laboratory, Riverside, CA

Table of Contents

List of Figures

List of Tables

Abstract

Šimůnek, J., Šejna, M., and M. Th. van Genuchten, The HYDRUS Software Package for Simulating Two- and Three-Dimensional Movement of Water, Heat, and Multiple Solutes in Variably-Saturated Media, User Manual, Version 1.0, PC Progress, Prague, Czech Republic, 2006.

This report documents version 1.0 of the **Graphical User Interface of HYDRUS**, a software package for simulating water, heat, and solute movement in two- and three- dimensional variably saturated media. The software package consists of the computational computer program, and the interactive graphics-based user interface. The HYDRUS program numerically solves the Richards equation for variably saturated water flow and advection-dispersion equations for both heat and solute transport. The flow equation incorporates a sink term to account for water uptake by plant roots. The heat transport equation considers transport due to conduction and convection with flowing water. The solute transport equations consider advective-dispersive transport in the liquid phase, as well as diffusion in the gaseous phase. The transport equations also include provisions for nonlinear nonequilibrium reactions between the solid and liquid phases, linear equilibrium reactions between the liquid and gaseous phases, zero-order production, and two first-order degradation reactions. In addition, physical nonequilibrium solute transport can be accounted for by assuming a two-region, dual-porosity type formulation which partitions the liquid phase into mobile and immobile regions. Attachment/detachment theory, including filtration theory, is additionally included to enable simulations of the transport of viruses, colloids, and/or bacteria.

HYDRUS may be used to analyze water and solute movement in unsaturated, partially saturated, or fully saturated porous media. The program can handle flow regions delineated by irregular boundaries. The flow region itself may be composed of nonuniform soils having an arbitrary degree of local anisotropy. Flow and transport can occur in the two-dimensional vertical or horizontal plane, a three-dimensional region exhibiting radial symmetry about the vertical axis, or a fully three-dimensional domain. The two-dimensional part of this program also includes a Marquardt-Levenberg type parameter optimization algorithm for inverse estimation of soil hydraulic and/or solute transport and reaction parameters from measured transient or steady-state data for two dimensional problems. Details of the various processes and features included in HYDRUS are provided in the **Technical Manual** [*Šimůnek et al.*, 2006].

The main program unit of the **HYDRUS Graphical User Interface** (GUI) defines the overall computational environment of the system. This main module controls execution of the program and determines which other optional modules are necessary for a particular application. The module contains a project manager and both the pre-processing and post-processing units. The pre-processing unit includes specification of all necessary parameters to successfully run the **HYDRUS** FORTRAN codes, grid generators for relatively simple rectangular and hexahedral transport domains, a grid generator for unstructured finite element meshes for complex two-dimensional domains, a small catalog of soil hydraulic properties, and a Rosetta Lite program for generating soil hydraulic properties from soil textural data. The post-processing unit consists of simple *x-y* graphics for graphical presentation of soil hydraulic properties, as well as such output as distributions versus time of a particular variable at selected observation points, and actual or cumulative water and solute fluxes across boundaries of a particular type. The post-processing unit also includes options

to present results of a particular simulation by means of contour maps, isolines, spectral maps, and velocity vectors, and/or by animation using both contour and spectral maps.

This report serves as a **User Manual** and reference document of the **Graphical User Interface** of the **HYDRUS** software package. Technical aspects such as governing equations and details about the invoked numerical techniques are documented in a separate **Technical Manual**.

Introduction to the HYDRUS Graphical User Interface

The past several decades or so has seen an explosion of increasingly sophisticated numerical models for simulating water flow and contaminant transport in the subsurface, including models dealing with one- and multi-dimensional flow and transport processes in the unsaturated or vadose zone between the soil surface and the ground water table. Even with an abundance of well-documented models now available, one major problem often preventing their optimal use is the extensive work required for data preparation, numerical grid design, and graphical presentation of the output results. Hence, the more widespread use of multi-dimensional models requires ways which make it easier to create, manipulate and display large data files, and which facilitate interactive data management. Introducing such techniques will free users from cumbersome manual data processing, and should enhance the efficiency in which programs are being implemented for a particular example. To avoid or simplify the preparation and management of relatively complex input data files for two- and three-dimensional applications, and to graphically display the final simulation results, we developed an interactive graphics-based user-friendly interface HYDRUS for the MS Windows 95, 98, NT, ME, and XP environments. The interface is connected directly to the computational codes. The current version 1.0 of the HYDRUS graphical user interface represents a complete rewrite of the version 2.0 of HYDRUS-2D, and expands its capabilities to three-dimensional problems.

In addition to information given in this user manual, extensive context-sensitive on-line help is made part of the graphical user interface (GUI). By holding the F1 button or clicking on the Help button while working in any window, the user obtains information about the window content. In addition, context-sensitive help is available in every module using the "SHIFT+F1" help button. In this mode, the mouse cursor changes to a help cursor (a combination arrow + question mark), which a user can use to select a particular object for which help is needed (e.g., a menu item, toolbar button, or other features). At that point, a help file will be displayed giving information about the item on which the user clicked. Except for the computational modules that are written in FORTRAN, the entire GUI is written in C++.

The HYDRUS Graphical User Interface (Fig. 1) is the main program unit defining the overall computational environment of the system. This main module controls execution of the program and determines which other optional modules are necessary for a particular application. The module contains a project manager and both the pre-processing and post-processing units. The pre-processing unit includes specification of all necessary parameters to successfully run the **HYDRUS** FORTRAN codes (modules H2D_CALC, H2D_CLCI, H2D_WETL and/or H3D_CALC), grid generators for relatively simple rectangular and hexahedral transport domains, a grid generator for unstructured finite element meshes appropriate for more complex two-dimensional domains, a small catalog of soil hydraulic properties, and a Rosetta Lite program for generating soil hydraulic properties from textural information. The post-processing unit consists of simple *x-y* graphs for graphical presentation of the soil hydraulic properties, distributions versus time of a particular variable at selected observation points, as well as actual or cumulative water and solute fluxes across boundaries of a particular type. The post-processing unit also includes options to present results of a simulation by means of contour maps, isolines, spectral maps, and velocity vectors, and/or by animation using both contour and spectral maps.
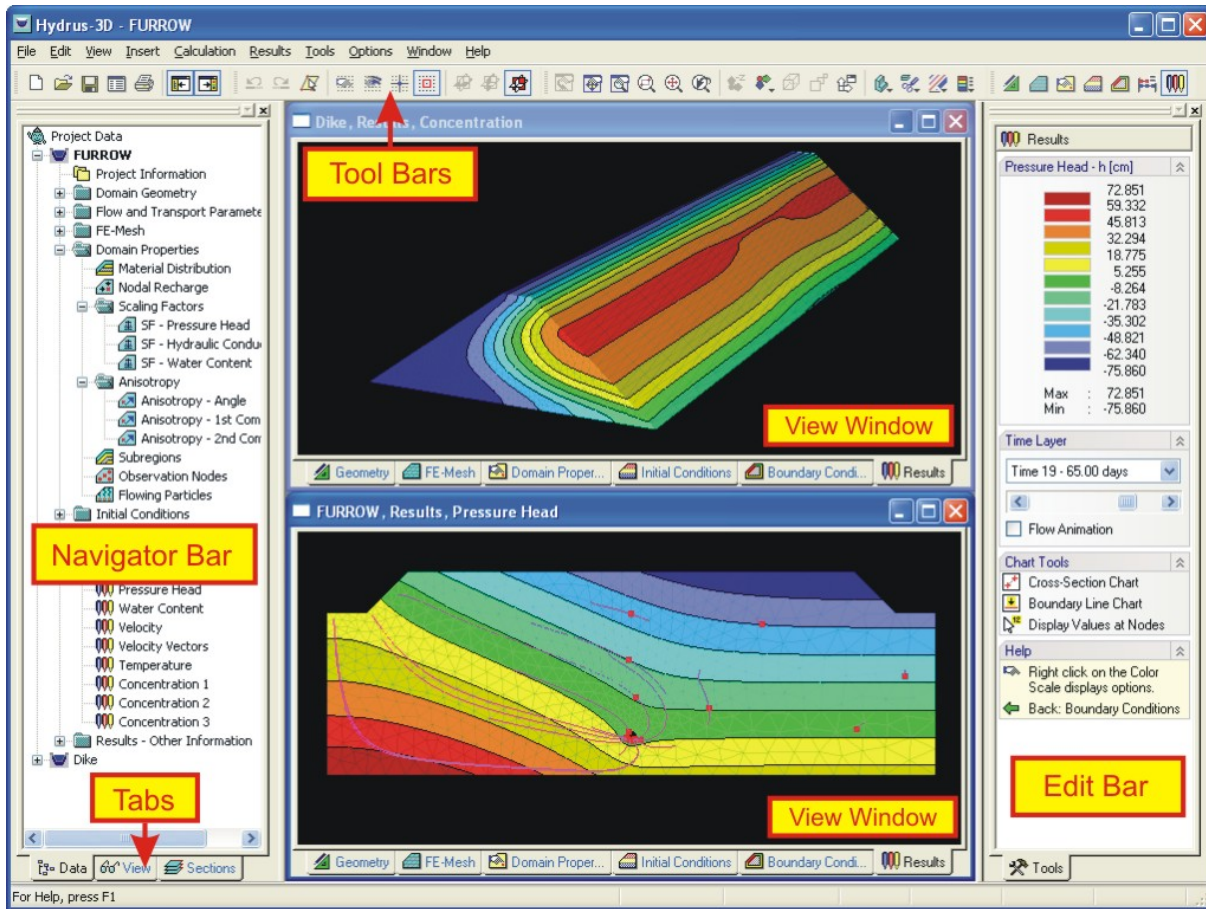
Figure 1. The HYDRUS Graphical User Interface (the main window).

Figure 1 shows the main window of the HYDRUS graphical user interface, including its main components such as the **Menu**, **Toolbars**, the **View Window**, the **Navigator Bar**, **Tabs**, and the **Edit Bar**. These terms will be used throughout this user manual. The text below provides a detailed description of all major components of the graphical user interface. At the end of this user manual a list is given of all commands accessible through the menu (Table 12), as well as a brief discussion of the action taken with particular commands (Table 13). More detailed descriptions are available through the on-line help.

Work for a new project should begin by opening the Project Manager (see Chapter 1), and giving a name and brief description to the new project. Next the **Geometry Information** dialog Window (Figs. 6 and 7) appears (this window can be also selected from the Pre-processing Menu). From this point on the program will navigate users through the entire process of entering input files. Users may either select particular commands from a menu, or allow the interface to lead them through the process of entering input data by selecting the Next button. Alternatively, clicking the Previous button will return users to the previous window. Pre- and post processing commands and processes are also sequentially listed on the **Data Tab** of the **Navigator Bar**. Green arrows on the **Edit Bar** always direct users to subsequent or previous input processes for a particular command. Many commands and processes can be alternatively accessed using either the **Toolbars** and **Menus**, or the **Navigator** and **Edit Bars**.

1. Project Manager and Data Management

A **Project Manager** (called by the command *File->Project Manager*, Figs. 2 and 3) is used to manage the data of existing projects, and helps to locate, open, copy, delete and/or rename desired projects or their input or output data. A **Project** represents any particular problem to be solved by HYDRUS. The project name, as well as a brief description of the project (Fig. 4), helps to locate a particular problem. Projects are represented by a file *project_name.h3d* that contains all input and output data. HYDRUS input files (used by the computational modules) are extracted from the *project_name.h3d* file into a working subdirectory; output data created by the calculation module are sent into the same folder. When saving a project, output files (created by the computational modules) are also included into the *project_name.h3d* file. The input and output files can be either permanently kept in the external working directory, or are stored in this folder only during calculations (Fig. 4, the radio buttons *Temporary – is deleted after closing the project* and *Permanent – result files are kept in this directory*). The location of the external working directory is specified in the **Program Options** dialog window (Fig. 92).
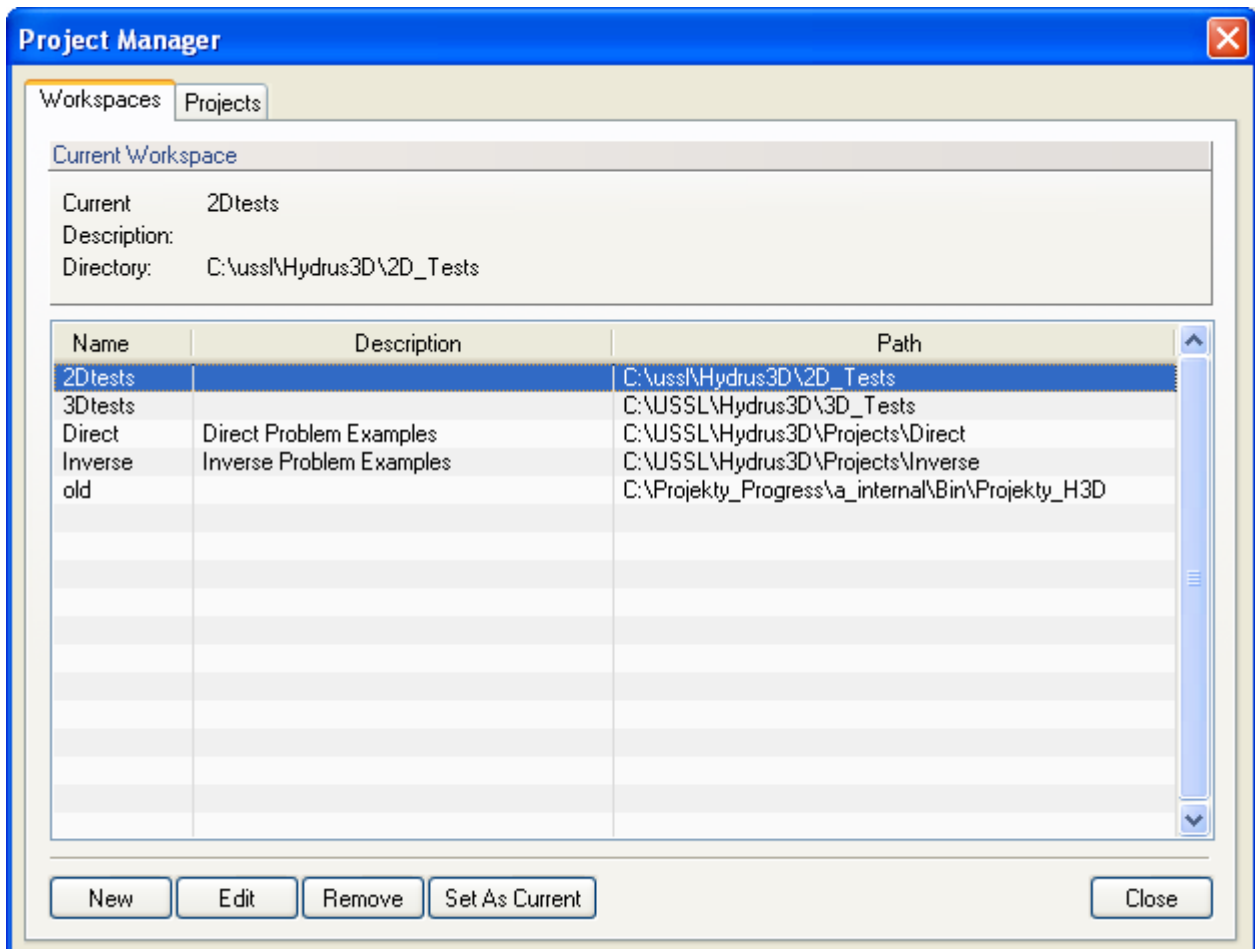
Figure 2. The project Manager with the Workspaces tab.
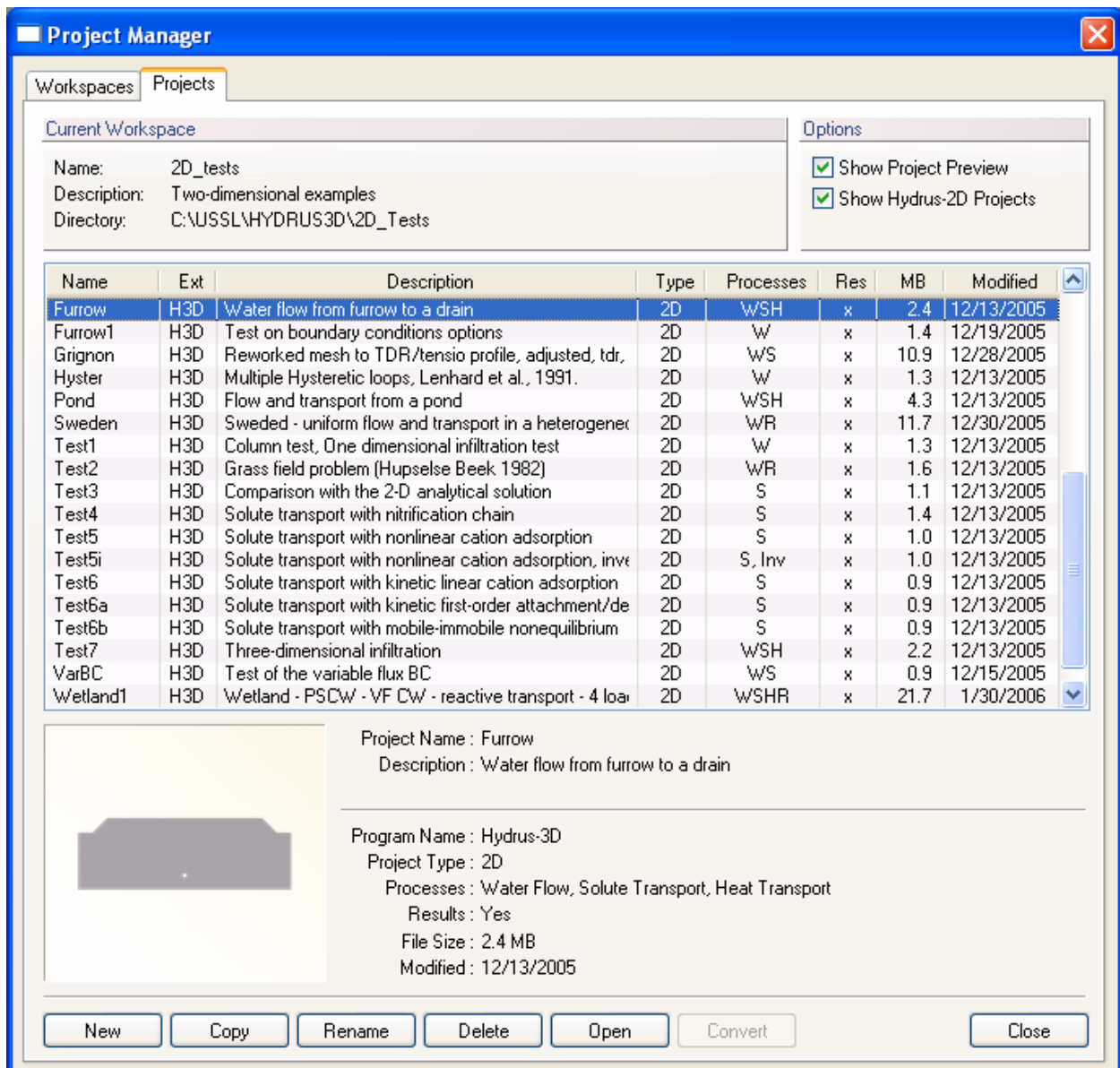
Figure 3. The Project Manager with the Projects tab.

The **Project Manager** gives users considerable freedom in organizing their projects. The projects are grouped into **Workspaces** (Fig. 2), which can be placed anywhere in accessible memory (i.e., on local and/or network hard drives). **Workspaces** serve to organize projects into logical groups defined by a user. Each **Workspace** has its own *name*, *description*, and *pathway* (Figs. 2 and 5). A **Workspace** can be any existing accessible subdirectory (folder). HYDRUS is installed together with two default workspaces, *2D_Tests* and *3D_Tests*, which are located in the HYDRUS3D folder. The *2D_Tests* and *3D_Tests* workspaces contain test examples for two- and three-dimensional problems, respectively. We suggest that users create their own workspaces (e.g., the *My_2D_Direct*, *My_2D_Inverse*, and *My_3D_Direct* workspaces), and keep the provided examples intact for future reference. Projects can be copied with the **Project Manager** only within a particular workspace. Users can copy projects between workspaces using standard file managing software, such as

Windows Explorer. In that case one must copy only the *project_name.h3d* file. When temporary data are kept in the working directory (i.e., the radio button *Temporary – is deleted after closing the project*, Fig. 4), the working directory must be copied together with the *project_name.h3d* file.

In addition to a *Name* and a brief *Description* of a **Project**, the **Project Manager** also displays dimensions for a particular problem (*Type*), what *Processes* are involved (W – water flow, S – solute transport, T – heat transport, R – root water uptake, Inv – Inverse problem), the size of the project (MB), when the project was created (*Date*) and whether or not the *Results* exist (Fig. 3). The **Project Manager** can also display a preview of the **Project**'s geometry (see the check box *Show Project Preview* in Fig. 3). Commands of the **Project Manager** are listed in Table 1.

Table 1. Commands in the **Project Manager**.

Group	Command	Description
Workspace	New	Registers a new workspace in the Project Manager.
	Edit	Renames the selected workspace, and changes its description and/or location.
	Remove	Removes registration of a selected workspace from the Project Manager.
	Set As Current	Sets a selected workspace as the active workspace.
	Close	Closes the Project Manager.
Project	New	Creates a new project in the current workspace.
	Copy	Copies a selected project within the current workspace.
	Rename	Renames a selected project.
	Delete	Deletes a selected project.
	Open	Opens a selected project.
	Close	Closes the Project Manager.
	Convert	Converts HYDRUS-2D (version 2.x) projects to HYDRUS.

The commands **New** and **Rename** from the **Project Tab** of the **Project Manager** dialog window (Fig. 3) call the **Project Information** dialog window (Fig. 4), which contains the *Name* and *Description* of the project, as well as information about the **Workspace** (name, description, and pathway) to which the project belongs. It also contains information whether or not the input and output data are kept permanently in an external directory (the radio buttons *Temporary – is deleted after closing the project* and *Permanent – result files are kept in this directory*, Fig. 4).
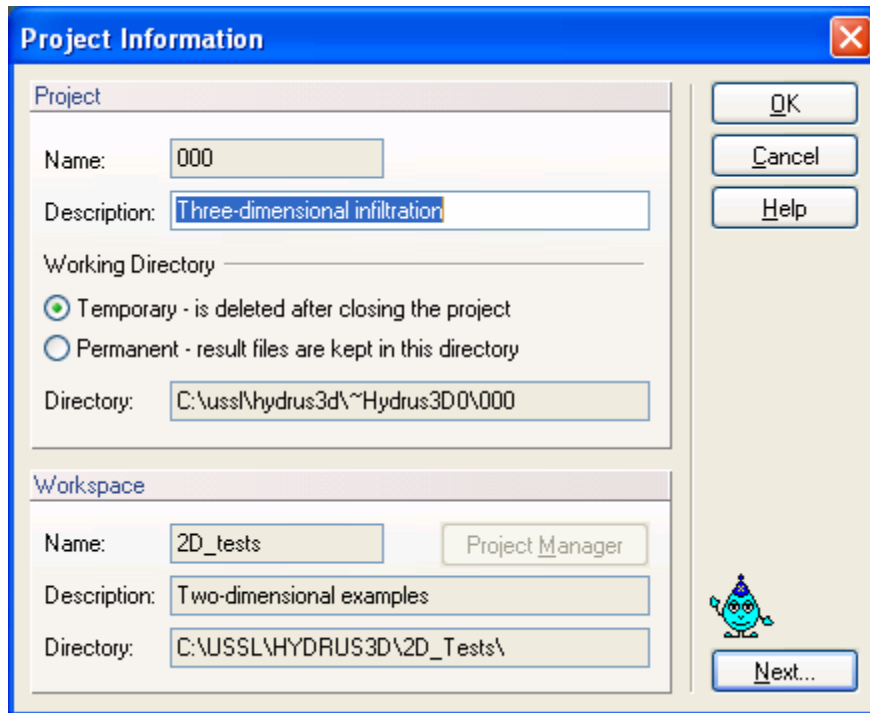
Figure 4. The Project Information dialog window.
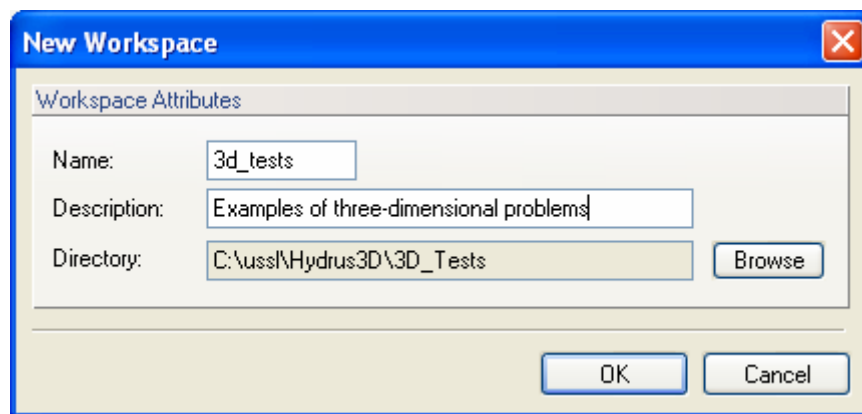
Figure 5. General description of the HYDRUS workspace.

Projects created by the previous version 2.0 of HYDRUS-2D can be imported into the current version of HYDRUS using two ways:

A. Individual projects can be converted using the command *File->Import and Export->Import HYDRUS-2D Project*. This is done by first creating a new **Project**, and then selecting the above command and browsing for the location of a project created with a previous version of HYDRUS-2D. The input data of the older project are then converted into the new HYDRUS format. Results of the older project can then be viewed using the new version of HYDRUS, while projects can be modified or recalculated as needed.

B. Multiple HYDRUS-2D projects can be converted simultaneously using the **Convert** command of the **Project Manager**. One first creates a HYDRUS workspace for a folder in which the HYDRUS-2D projects are located and selects the **Show HYDRUS-2D Projects** option at the **Project Tab** of the **Project Manager**. One then selects projects to be converted and clicks the **Convert** command. HYDRUS in this way creates HYDRUS projects and stores all input and output files in the **project_name.h3d** files.

Input data can be edited either using the HYDRUS graphical user interface (this modifies directly the *project_name.h3d* file) or the input data can be modified manually. In such case, HYDRUS input files need to be stored in the working external directory (sent there by the command *File->Import and Export->Export Data for HYDRUS Solver*), and then can be imported back into the HYDRUS *project_name.h3d* file using the command *File->Import and Export->Import Input Data from *.In Files*.

The **Working Directory** is a folder into which the program stores temporary data. Each open project has its own Working Directory, where the program stores, for example, input files for computational modules, and where computational modules write the output files. When saving a project, data from the Working Directory are copied into the main project file *project_name.h3d*. When the project is closed, the Working Directory is deleted. Only when a user selects the option "*Permanent – result files are kept in this directory*" (Fig. 4) is the Working Directory not deleted after closing the project, in which case the temporary data are not copied into the main project file.

2. Projects Geometry Information

HYDRUS can solve water flow, and solute and heat transport for two- and three-dimensional transport domains. Geometry type is selected in the **Geometry Information** dialog Window (Fig. 6 and 7). In this dialog window, users specify the **Type of Geometry**, the **Domain Definition**, the **Length Units**, and the size of the **Initial Workspace** (the approximate size of the transport domain).

Type of Geometry: In the first dialog window that a user encounters after creating a new project, he/she needs to specify whether the flow and transport problem occurs in a two- or three-dimensional transport domain. Two-dimensional flow and transport can occur in a horizontal or vertical plane, or in an axisymmetrical quasi-three-dimensional transport domain. When a three-dimensional axisymmetrical system is selected, the *z*-coordinate must coincide with the vertical axis of symmetry. A typical example of the selected 2D or 3D geometry is shown in the preview part of the dialog window.
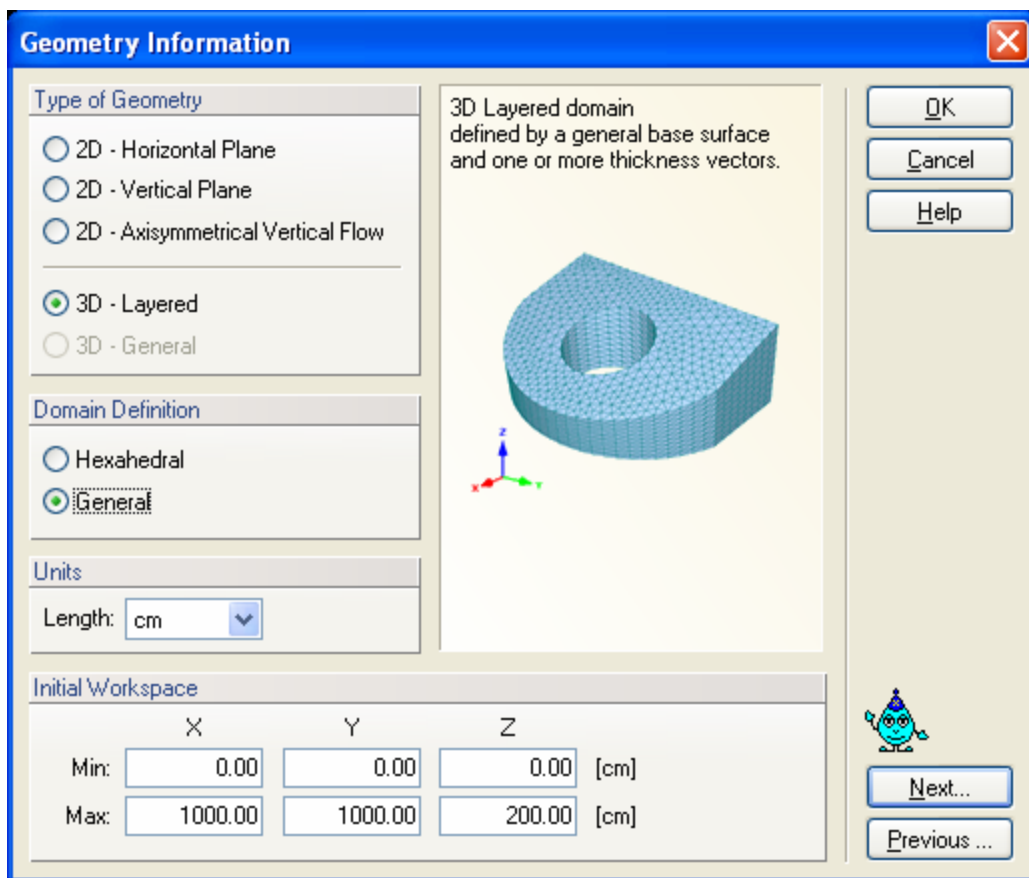
Figure 6. The Geometry Information dialog window (with 3D preview).
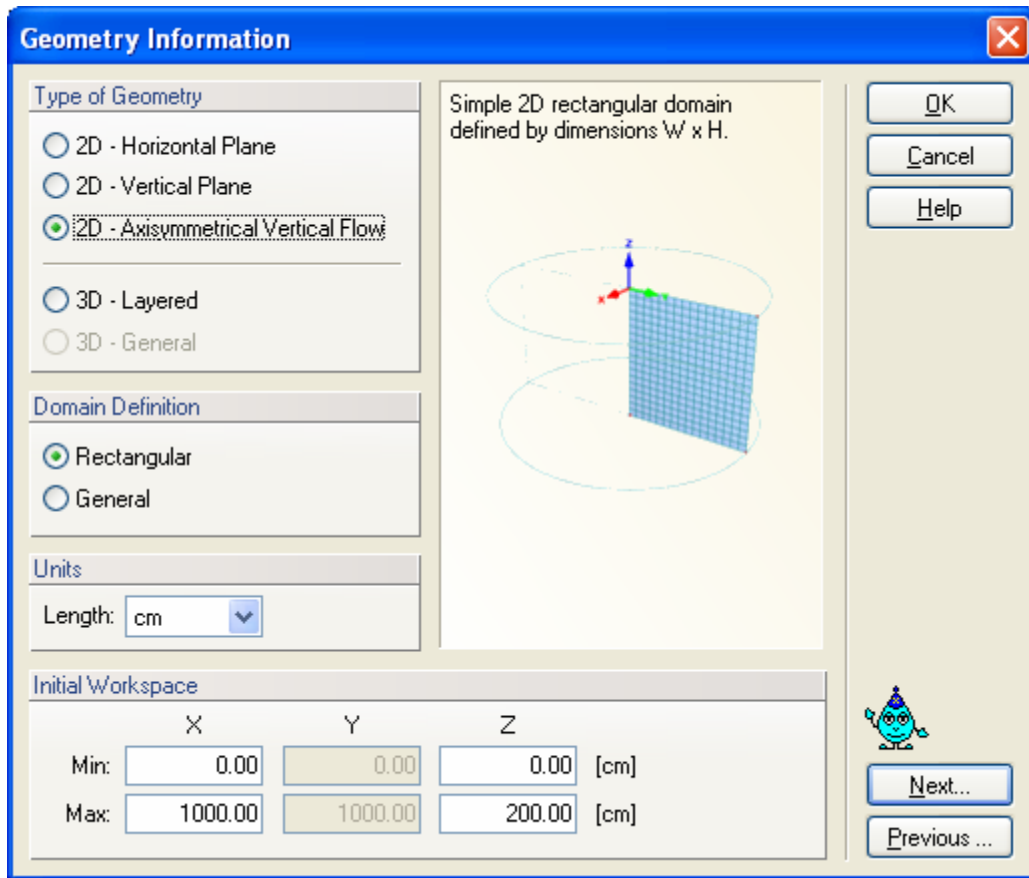
Figure 7. The Geometry Information dialog window (with 2D axisymmetrical preview).

Domain Definition: This section allows a user to choose between a simple geometry having a structured finite element mesh, or a more general geometry having an unstructured finite element mesh. Only simple geometries may be used depending on the authorization. Simple rectangular domains are defined by three straight lines, one at the bottom of the domain and two at the sides, whereas the upper boundary may or may not be straight. Nodes along the upper boundary line may in that case have variable *x*- and *z*-coordinates. However, the lower boundary line must always be horizontal (or have a specified slope), while the left and right boundary lines must be vertical. The flow region is then discretized into a structured triangular mesh. Hexahedral domains must have similar properties as rectangular domains, i.e., vertical planes at the sides, a horizontal (or with a specified slope) plane at the bottom boundary, and with only the upper boundary not needing to be a plane. Examples of simple rectangular and general two-dimensional geometries are shown in Figure 8. An example of a simple hexahedral three-dimensional geometry is given in Figure 9.
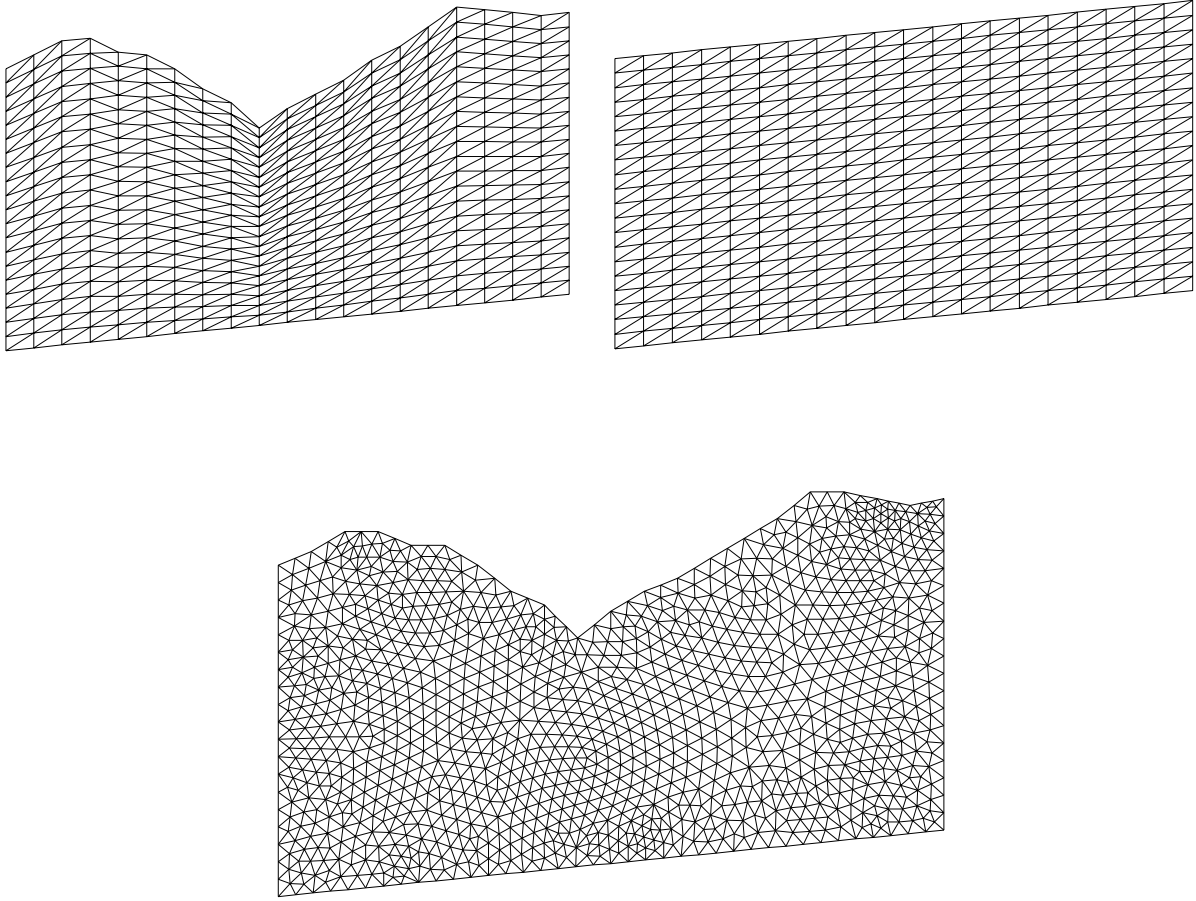
Figure 8. Examples of rectangular (top) and general (bottom) two-dimensional geometries.
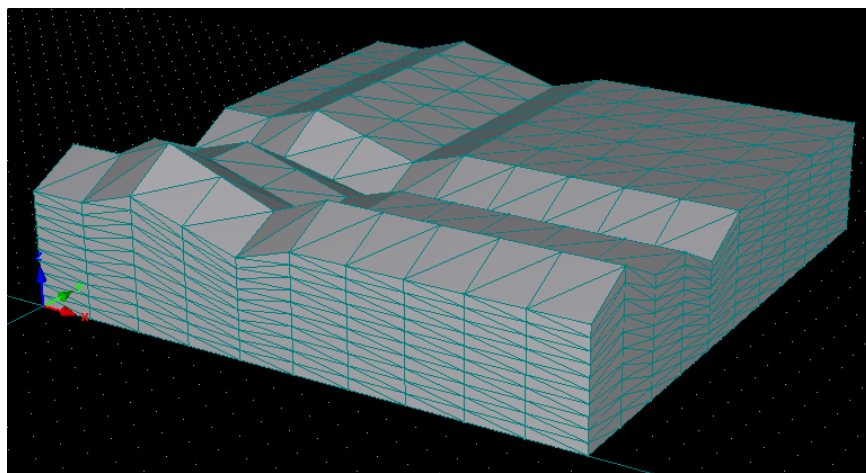
Figure 9. Example of a hexahedral three-dimensional geometry.

The simple geometries are defined in the *Rectangular* (Fig. 10) or *Hexahedral Domain Definition* (Fig. 11) dialog windows for two-dimensional and three-dimensional problems, respectively. In each of these windows, users need to specify the vertical and horizontal dimensions of the transport domain, as well as a possible slope of the base of the domain in different directions (if applicable; α is in the *x*-direction and β is in the *y*-direction). The preview in the middle of the dialog window of a simple example showing all geometry parameters should help users in specifying their desired transport domain (*Dimensions* and *Slope*).
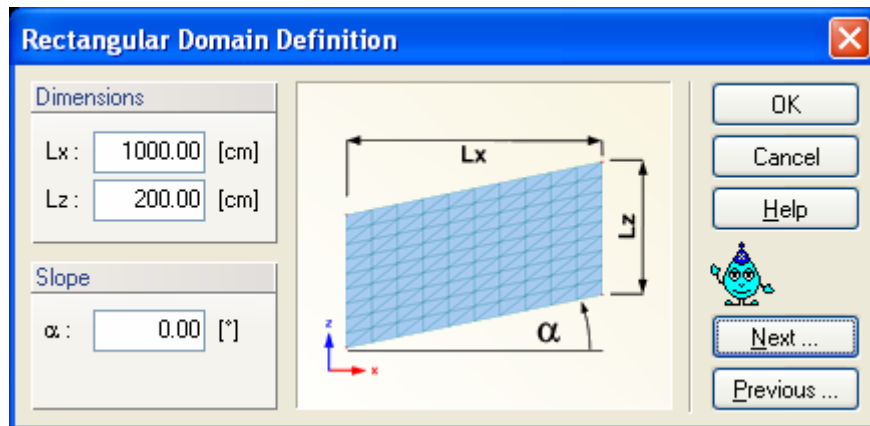
Figure 10. The Rectangular Domain Definition dialog window.
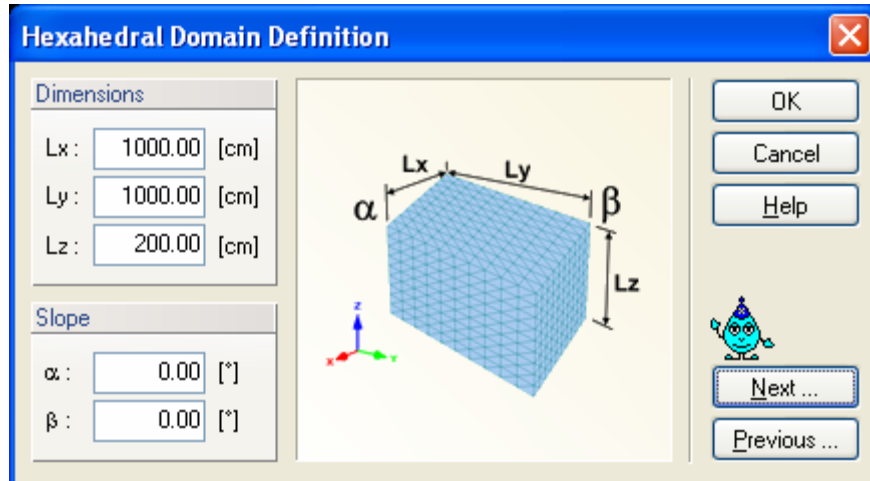
Figure 11. The Hexahedral Domain Definition dialog window.

In the **Geometry Information** dialog Window (Figs. 6 and 7) users also select the geometry **Units** to be used throughout the application (mm, cm, m) and the size of the **Initial Workspace**. When units are changed during specification or after reading the input data, then all input variables are automatically converted into the new units.

Initial Workspace: This part of the dialog allows users to define the initial dimensions of the graphical view window.

27

There are three types of three-dimensional transport domains (Solids, see also Section 4.4) depending upon the selection made in the Geometry Information dialog window (Fig. 6 and 7):

- **3D-Layered – Hexahedral:** This type of solid has a **Hexahedral Shape** and is defined by its basic dimensions. The base can have a certain slope in the X and Y dimensions.

- **3D-Layered – General:** This type of solid is defined by the **Base Surface** (see Section 4.2) and one or more **Thickness Vectors** (see Section 4.5).

- **3D-General:** This type of solid is defined using a set of surfaces that form its boundaries. This selection is not possible in the current version of HYDRUS.

3. Flow Parameters

3.1. *Main Processes*

In the **Main Processes** dialog window (Fig. 12), users specify the processes to be simulated, i.e., water flow, multiple solute transport, heat transport, and/or root water uptake.

The program automatically considers transient water flow when the "water flow" option is selected. Otherwise the code tries to calculate steady-state flow from the specified initial and boundary conditions. The success of such calculations depends on the complexity and/or nonlinearity of the problem. If unsuccessful, then a model run with constant boundary conditions and long simulation time may be required.

If the solute transport, heat transport or root water uptake options originally considered in an existing project are switched off by the user, the program issues a warning that all data related to these processes will be lost. If this loss is undesirable we recommend that users first copy the input data of the current project to a new project before switching off the solute transport, heat transport and/or root water uptake options.

For two-dimensional problems a user can also select if a **Direct** or **Inverse problem** is to be solved. Inverse problems involve the estimation of selected parameters from available experimental data.
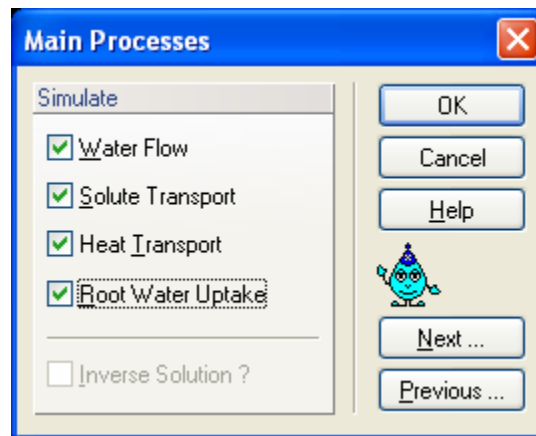
Figure 12. The Main Processes dialog window.

3.2. *Inverse Solution*

HYDRUS implement a Marquardt-Levenberg type parameter estimation technique (Šimůnek and Hopmans, 2002) for inverse estimation of soil hydraulic (Hopmans et al., 2002) and/or solute transport and reaction (Šimůnek et al., 2002) parameters from measured transient or steady-state flow and/or transport data. The **Inverse Solution** dialog window (Fig. 13) appears only when the **Inverse Problem** in the **Main processes** dialog window (Fig. 12) is selected. Users select which parameters (the soil hydraulic, solute transport and reaction, and/or heat transport parameters) are to be optimized (**Estimate …**) from the specified experimental data.

One also selects the method of **Weighting of Inversion Data** in the objective function. Users can choose between no weighting, weighting by mean ratios, or weighting by standard deviations. When no weighting is selected, one needs to supply weights for particular data points in the **Data for Inverse Solution** dialog window (Fig. 14). When weighting by mean ratio or weighting by standard deviation is selected, then the code calculates either the means or the standard deviations of the different data sets (e.g., water contents, pressure heads, concentrations, …) and adjusts the weights proportionally. These internal weights can still be multiplied by weights from the **Data for Inverse Solution** dialog window (Fig. 14).
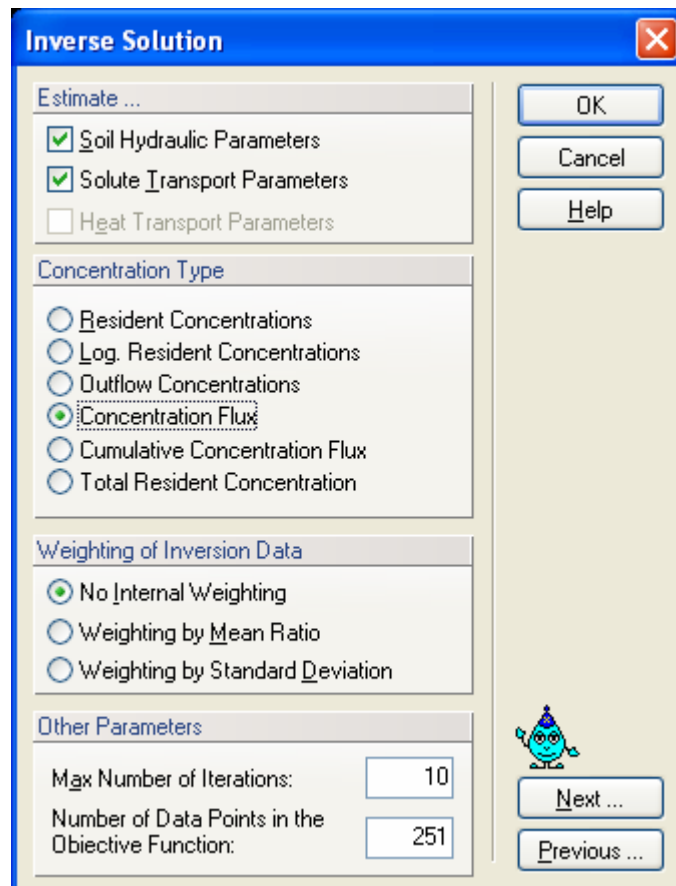
Figure 13. The Inverse Solution dialog window.

The objective function for the inverse estimation of solute transport parameter can be defined using different types of concentrations. Available **Concentration Types** are: a) the resident concentration in the liquid phase, b) a log-transformation of the resident concentration in the liquid phase, c) the outflow (flux) concentration, d) the solute concentration flux, e) the cumulative concentration (solute) flux, and f) the total resident concentration. The total resident concentration includes concentrations in the sorbed and nonequilibrium phases.

The maximum number of iterations for the inverse solution is also specified in this dialog window. If one selects zero number of iterations then only the direct simulation is carried out. However, users can still enter measured data, in which case the code compares results of the direct simulation with the measured data.
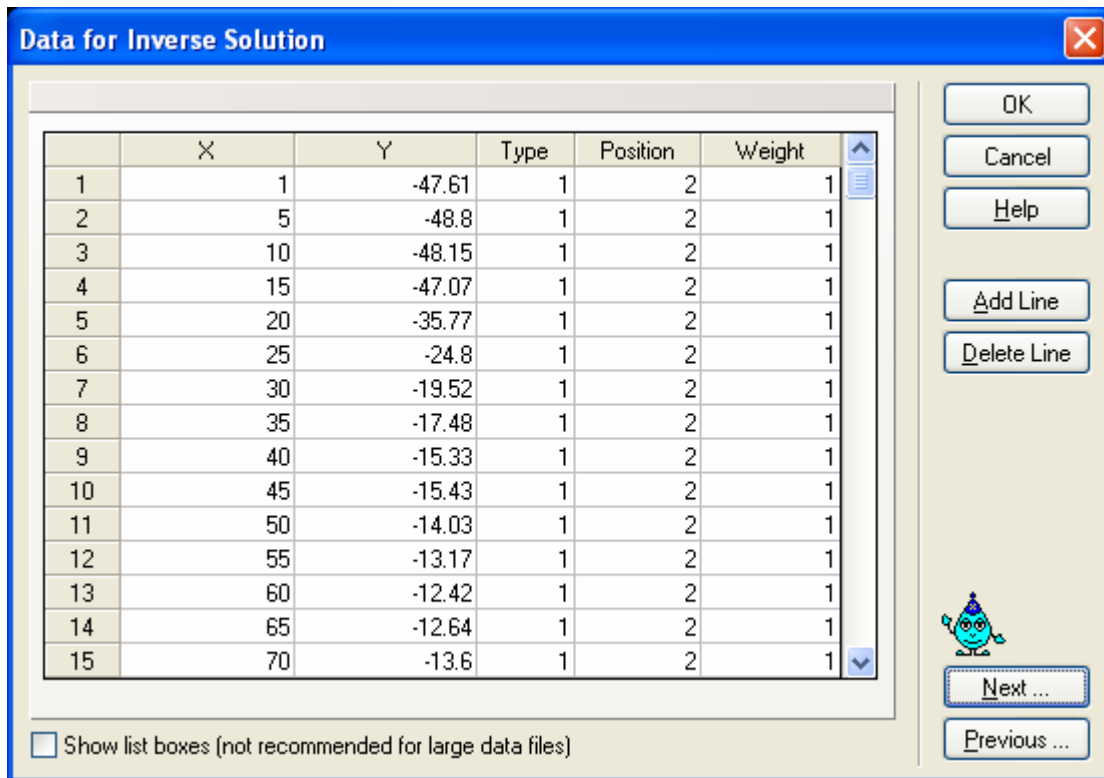
Figure 14. The Data for Inverse Solution dialog window.

In the table **Data for Inverse Solution** (Fig. 14) one specifies the measured data that will be analyzed during the parameter optimization process. Many different types of data can be used to define the objective function that will be minimized for this purpose. How the values in the **X** and **Y** columns are interpreted depends on the **Type** and **Position** values. **Weight** is the weight associated with a particular data point. The following information can be included into the objective function:

Table 2. Data Types for the objective function (Inverse Problem).

Type	Data Point
0	Cumulative boundary fluxes across a specified boundary
1	Pressure head measurements at selected observation point(s)
2	Water content measurements at selected observation point(s)
3	Boundary flux across a specified boundary
4	Concentration or temperature measurements at observation point(s)
5	$h(\theta)$ measurements; soil water retention data
6	$K(h)$ measurements; hydraulic conductivity data point
7	Prior knowledge of parameter α
8	Prior knowledge of parameter n
9	Prior knowledge of parameter θ_r
10	Prior knowledge of parameter θ_s
11	Prior knowledge of parameter K_s

Depending upon the value of parameter **Type**, the first column (X) contains the following information:

Table 3. Definition of the column X in Fig. 14 based on Data Type (Inverse Problem).

X	Type
Time	0, 1, 2, 3, 4
Pressure head	5, 6
Dummy variable	7, 8, 9, 10, 11

Depending upon the value of parameter **Type**, the second (Y) and fourth (Position) columns contain the following information:

Table 4. Definition of the column *Y* in Fig. 14 based on Data Type (Inverse Problem).

Y	Type	Position
Cumulative boundary flux across a specified boundary	0	Code for the specified boundary[*]
Pressure head	1	Observation node number
Water content	2	Observation node number
Averaged water content of the entire flow domain	2	0
Averaged water content of the subregion	2	Negative subregion number
Concentrations/temperatures	4	Observation node number
Concentrations for the second solute	4	Negative observation node number
Total amount of solute in the entire flow domain	4	0
Boundary flux across a specified boundary	3	Code for the specified boundary[*]
Water content	5	Material number
Hydraulic conductivity	6	Material number
Particular parameter	7-11	Material number

[*]1 – constant pressure head or flux boundary; 2 – seepage face; 3 – variable pressure head or flux boundary 1; 4 – atmospheric boundary; 5 – drains; 6 – free or deep drainage boundary; 7, 8, and 9 – variable pressure head or flux boundaries 2, 3, or 4, respectively.

3.3. *Time Information*

The **Time Information** dialog window (Fig. 15) contains information associated with the **Time Discretization**, the **Time Units**, and the implementation of **Boundary Conditions**.
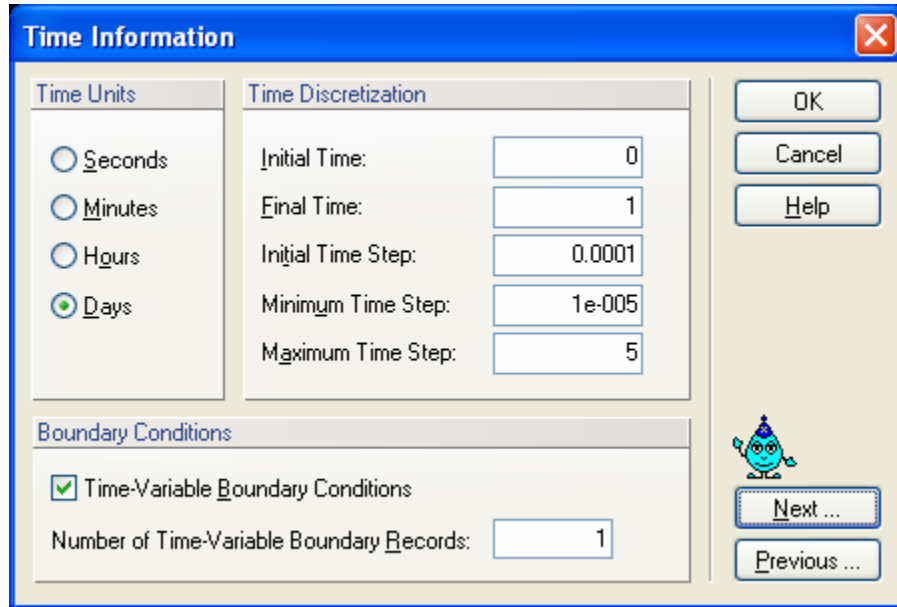
Figure 15. The Time Information dialog window.

Table 5. Time Information variables.

Time Units	Time units [T] to be used throughout the application (days, hours, min, sec). When units are changed during or after data entry, then all input variables are converted automatically into the new units.
Initial Time	Starting time [T] of the calculation.
Final Time	Final time [T] of the calculation.
Initial Time Step	Initial time increment, dt [T]. The initial time step should be a function of the type of problem being solved. Problems with high pressure gradients (e.g., infiltration into an initially dry soil) and for soils with highly nonlinear soil hydraulic properties require relatively small initial time steps. The initial time step is used at the beginning of the simulation or whenever boundary conditions are substantially changed (e.g., the water flux changes by 25% or more).
Minimum Time Step	Minimum permitted value of the time increment, dt_{min} [T].
Maximum Time Step	Maximum permitted value of the time increment, dt_{max} [T].

Boundary Conditions

Time-Variable Boundary Condition - The number of time-dependent boundary records and time-dependent boundary conditions must be specified when this box is checked. The boundary conditions otherwise are assumed to be constant in time.

3.4. *Output Information*

The **Output Information** dialog window (Fig. 16) contains information governing output from the computational module of HYDRUS.
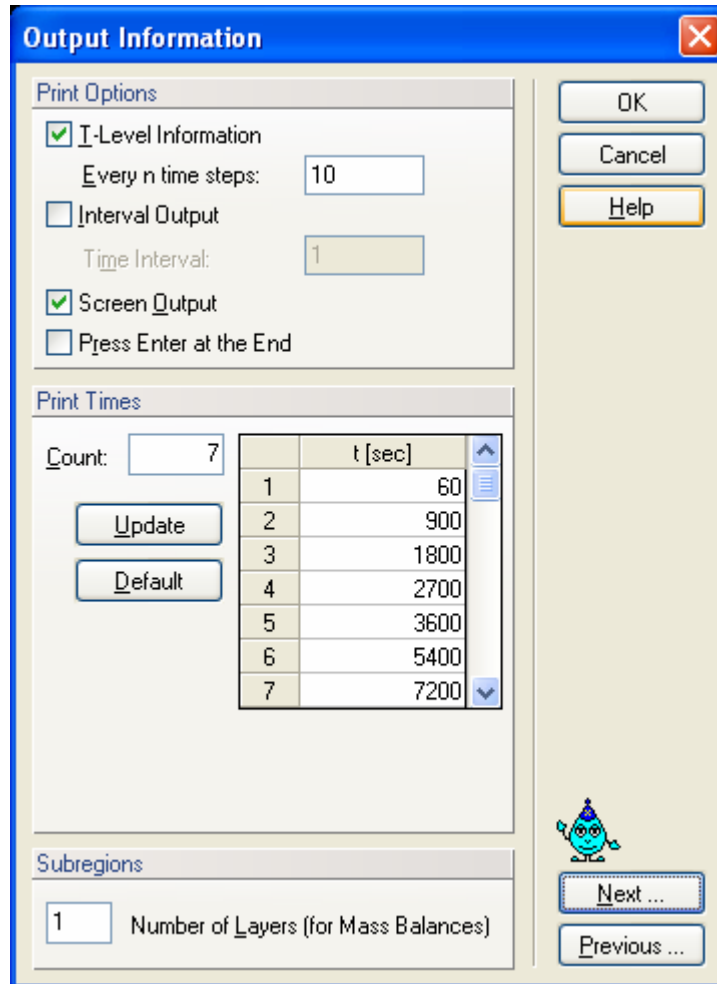
Figure 16. The Output Information dialog window.

In the **Print Options** part of the dialog window, one decides whether certain information concerning mean pressure heads and concentrations, mean water and solute fluxes, cumulative water and solute fluxes, and time and iteration information is printed at each time step (*T-Level Information*), after *n* time steps (*Every n time steps*), at a certain defined time interval (*Interval Output*), or if the information is sent to the screen during the calculations (*Screen Output*). When the simulation ends users are by default asked to hit the Enter key of the keyboard to return to the GUI from the computational window. This action can be disabled by unchecking the **Hit Enter at the End** check box.

T-Level Information This check box decides whether certain information concerning mean pressure heads and concentrations, mean water and solute fluxes, cumulative water and solute fluxes, and time and iteration information, are to be printed at each time step, after *n* time steps, or only at preselected times (**Print Times**) or **Time Intervals**.

Interval Output Users can specify whether or not information concerning mean pressure heads and concentrations, mean water and solute fluxes, cumulative water and solute fluxes, and time and iteration information is to be printed at a regular **Time Interval**.

Screen Output Check box to decide whether or not information about the simulation run is to be printed to the screen during execution of the HYDRUS computational code. We recommend to check this box for direct problems, but not for inverse problems.

In the **Print Times** part of the dialog window one specifies the number of Print Times (**Count**) at which detailed information about the pressure heads, water contents, concentrations, temperatures, fluxes, and the soil water and solute balances will be printed. Clicking on the **Default** command button will cause the print times to be distributed evenly between the initial and final time.

Finally, in the **Subregions** part one selects the number of regions for which a mass balance will be evaluated and printed to the Balance.out output file.

3.5. *Iteration Criteria*

The **Iteration Criteria** dialog window (Fig. 17) contains information related to the iterative process that is used to solve the Richards equation. Because of the nonlinear nature of the Richards equation, an iterative process must be used to obtain solutions of the global matrix equation at each new time step. For each iteration a system of linearized algebraic equations is first derived and then solved using either Gaussian elimination or the conjugate gradient method. After solving the matrix equation, the coefficients are re-evaluated using this solution, and the new equations are again solved. The iterative process continues until a satisfactory degree of convergence is obtained, i.e., until for all nodes in the saturated (unsaturated) region the absolute change in pressure head (water content) between two successive iterations becomes less than some small value determined by the imposed absolute **Pressure Head** (or **Water Content**) **Tolerance**. The first estimate (at zero iteration) of the unknown pressure heads at each time step is obtained by extrapolation from the pressure head values at the previous two time levels.
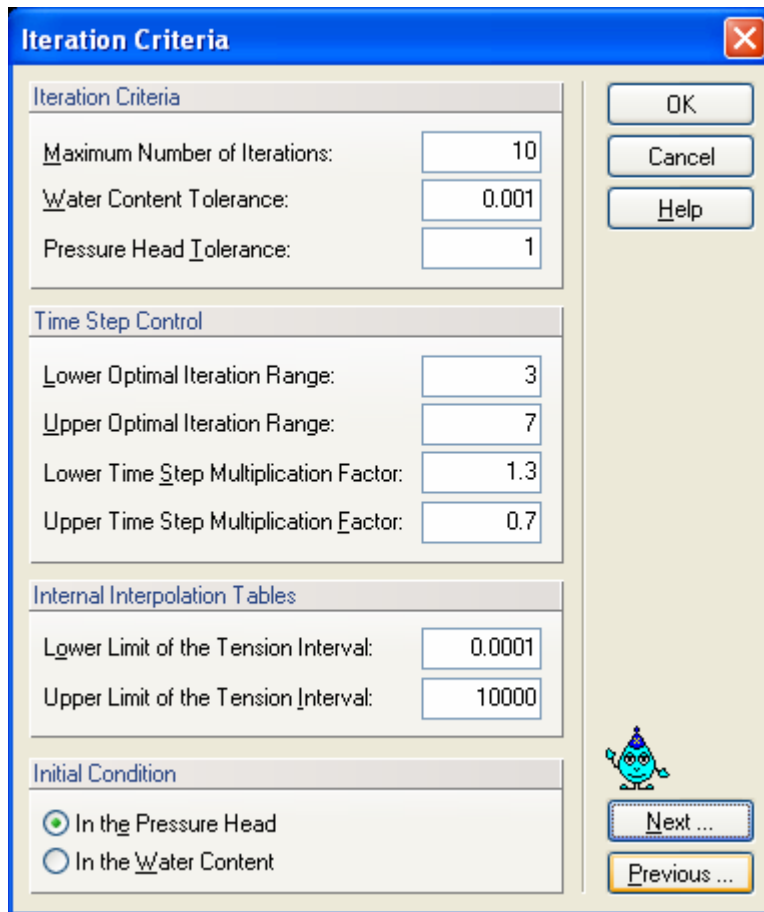
Figure 17. The Iteration Criteria dialog window.

In the **Iteration Criteria** part of the dialog window, one specifies the maximum number of iterations during one time step, and the water content and pressure head precision tolerances.

Max. Number of Iterations Maximum number of iterations allowed during any time step while solving the nonlinear Richards equation using a modified Picard method. The recommended and default value is 10.

Water Content Tolerance Absolute water content tolerance for nodes in the unsaturated part of the flow region [-]. When the water contents between two successive iterations during a particular time step change less than this parameter, the iterative process stops and the numerical solution proceeds to the new time step. Its recommended and default value is 0.001.

Pressure Head Tolerance Absolute pressure head tolerance for nodes in the saturated part of the flow region [L]. When the pressure heads between two successive iterations during a particular time step change less than this parameter, the iterative process stops and the numerical solution proceeds to the new time step. Its recommended and default value is 1 cm.

Information specified in the **Time Step Control** part of the dialog window is related to the automatic adjustment of the time step during calculations. Four different time discretizations are introduced in HYDRUS: (1) time discretizations associated with the numerical solution, (2) time discretizations associated with implementation of boundary conditions, (3) time discretizations associated with data points used in the inverse problem, and (4) time discretizations which provide printed output of the simulation results (e.g., nodal values of dependent variables, water and solute mass balance components, and other information about the flow regime).

Discretizations 2, 3, and 4 are mutually independent; they generally involve variable time steps as described in the input data file (**Time-Variable Boundary Conditions**, Fig. 31 and **Output Information,** Fig. 16). Discretization 1 starts with a prescribed initial time increment, Δt. This time increment is automatically adjusted at each time level according to the following rules:

    a. Discretization 1 must coincide with time values resulting from discretizations 2, 3, and 4.
    b. Time increments cannot become less than a preselected minimum time step, Δt_{min}, nor exceed a maximum time step, Δt_{max} (i.e., $\Delta t_{min} \le \Delta t \le \Delta t_{max}$).
    c. If, during a particular time step, the number of iterations necessary to reach convergence is ≤3, the time increment for the next time step is increased by multiplying Δt with a predetermined constant >1 (usually between 1.1 and 1.5). If the number of iterations is ≥7, Δt for the next time level is multiplied by a constant <1 (usually between 0.3 and 0.9).
    d. If, during a particular time step, the number of iterations at any time level becomes greater than a prescribed maximum (usually between 10 and 50), the iterative process for that time level is terminated. The time step is subsequently reset to $\Delta t/3$, and the iterative process restarted.

We note that the selection of optimal time steps, Δt, during execution is also influenced by the adopted solution scheme for solute transport.

Table 6. Time Step Control variables.

Lower Optimal Iteration Range	When the number of iterations necessary to reach convergence for water flow is less than this number, the time step is multiplied by the lower time step multiplication factor (the time step is increased). Recommended and default value is 3.
Upper Optimal Iteration Range	When the number of iterations necessary to reach convergence for water flow is higher than this number, the time step is multiplied by the upper time step multiplication factor (the time step is decreased). Recommended and default value is 7.
Lower Time Step Multiplication Factor	If the number of iterations necessary to reach convergence for water flow is less than the lower optimal iteration range, the time step is multiplied by this number (time step is increased). Recommended and default value is 1.3.
Upper Time Step Multiplication Factor	If the number of iterations necessary to reach convergence for water flow is higher than the upper optimal iteration range, the time step is multiplied by this number (time step is decreased). Recommended and default value is 0.7.

Internal Interpolation Tables. At the beginning of a numerical simulation, HYDRUS generates for each soil type in the flow domain a table of water contents, hydraulic conductivities, and specific water capacities from the specified set of hydraulic parameters. Values of the hydraulic properties are then computed during the iterative solution process using linear interpolation between entries in the table. If the pressure head h at some node falls outside the prescribed interval (ha, hb), the hydraulic characteristics at that node are evaluated directly from the hydraulic functions (i.e., without interpolation). The above interpolation technique was found to be much faster computationally than direct evaluation of the hydraulic functions over the entire range of pressure heads. Interpolation using tables can be avoided by setting ha and hb both to zero. Then the soil hydraulic properties are always evaluated directly from the hydraulic functions (i.e., without interpolation). Output graphs of the soil hydraulic properties will be given also for the interval (ha, hb).

Lower limit of the tension interval	Absolute value of the lower limit [L] of the pressure head interval for which a table of hydraulic properties will be generated internally for each material.
Upper limit of the tension interval	Upper value of the lower limit [L] of the pressure head interval for which a table of hydraulic properties will be generated internally for each material.

Finally, in the **Initial Conditions** part of the dialog window, a user specifies whether the initial conditions for the water flow calculations are to be specified in terms of the pressure head or water content.

3.6. *Soil Hydraulic Model*

In the **Soil Hydraulic Model** dialog window (Fig. 18) users select the **Hydraulic Model** to be used to describe the soil hydraulic properties, and specify whether or not **Hysteresis** is to be considered during the calculations.
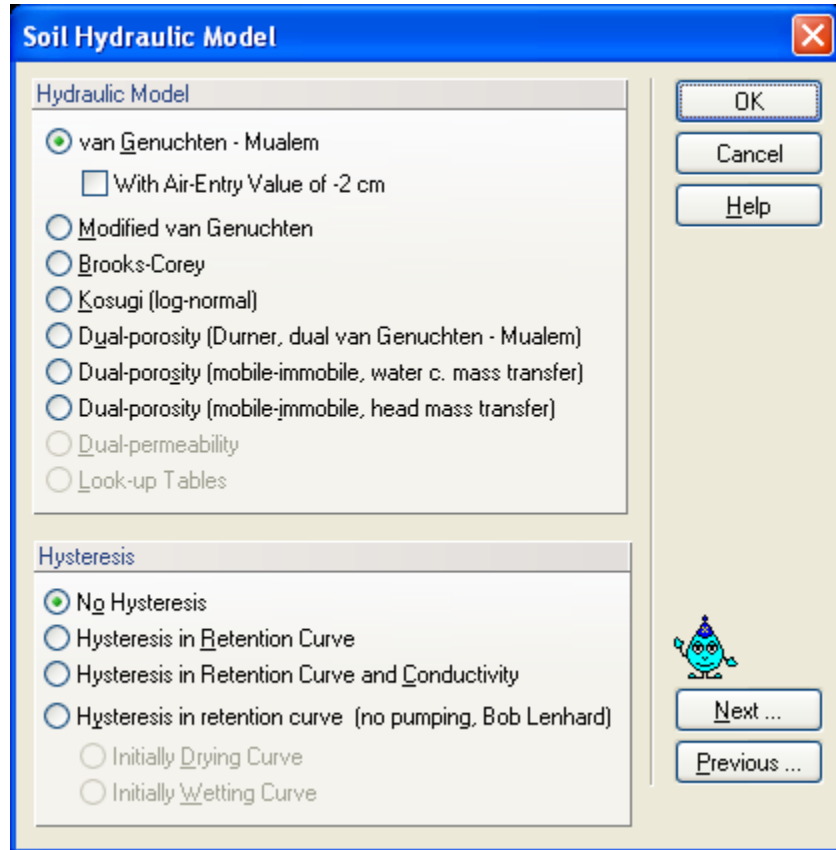
Figure 18. The Soil Hydraulic Model dialog window.

Hydraulic Model: The code allows users to select six types of models for the soil hydraulic properties: a) the van Genuchten-Mualem model [*van Genuchten*, 1980], b) the van Genuchten-Mualem model with an air-entry value of -2 cm, c) the modified van Genuchten type equations [*Vogel and Cislerova*, 1988], d) the equations of *Brooks and Corey* [1964], e) the lognormal distribution model of *Kosugi* [1996], and f) a dual-porosity model [*Durner*, 1994]. Additionally, user can select two dual-porosity nonequilibrium flow models with mass transfer between the mobile and immobile zones assumed to be proportional to either f) the water content or g) the pressure head. For a detailed description of these models, see the technical manual of HYDRUS. Two other approaches (a dual-permeability model and look-up tables) are not available in the current version of HYDRUS.

Hysteresis: When the van Genuchten model is used, either a) a non-hysteretic description (*No Hysteresis*), b) a hysteretic description only in the retention curve (*Hysteresis in Retention Curve*), or c) hysteretic descriptions in both the retention curve and the hydraulic conductivity curve (*Hysteresis in Retention Curve and Conductivity*) can be used. When hysteresis in the soil hydraulic properties is assumed, users must specify whether the initial condition is associated with the main wetting (**Initially Wetting Curve**) or main drying (**Initially Drying Curve**) retention curve. The HYDRUS code incorporates hysteresis by using the empirical model introduced by *Scott et al.* [1983]. This model was also employed by *Kool and Parker* [1987], who modified the formulation to account for air entrapment. While relatively simple to implement, the above model has been found to suffer sometimes from a so-called pumping effect, in which the hysteresis loops can move to physically unrealistic parts of the retention function. As an alternative, we also incorporated in HYDRUS the hysteresis model of *Lenhard et al.* [1991] and *Lenhard and Parker* [1992] that eliminates pumping by keeping track of historical reversal points (*Hysteresis in Retention Curve, no pumping, Bob Lenhard*).

3.7. *Water Flow Parameters*

Parameters for the soil hydraulic models are specified in the **Water Flow Parameters** dialog window (Fig. 19).

In all models (i.e., *Brooks and Corey*, 1964; *van Genuchten*, 1980; *Vogel and Cislerova*, 1988; *Kosugi*, 1996, and *Durner*, 1994), θ_r (Qr) and θ_s (Qs) denote the residual and saturated water contents, respectively; K_s (Ks) $[LT^{-1}]$ is the saturated hydraulic conductivity, and l is a pore-connectivity parameter. The parameters α (Alpha) $[L^{-1}]$ and n [-] are empirical coefficients affecting the shape of the hydraulic functions.

The modified van Genuchten model has four additional parameters: θ_a (Qa) – a water content smaller or equal to θ_r, θ_m (Qm) – a water content larger or equal to θ_s, K_k (Kk) $[LT^{-1}]$ – the unsaturated hydraulic conductivity at water content θ_k, and θ_k (Qk) – the water content associated with K_k.
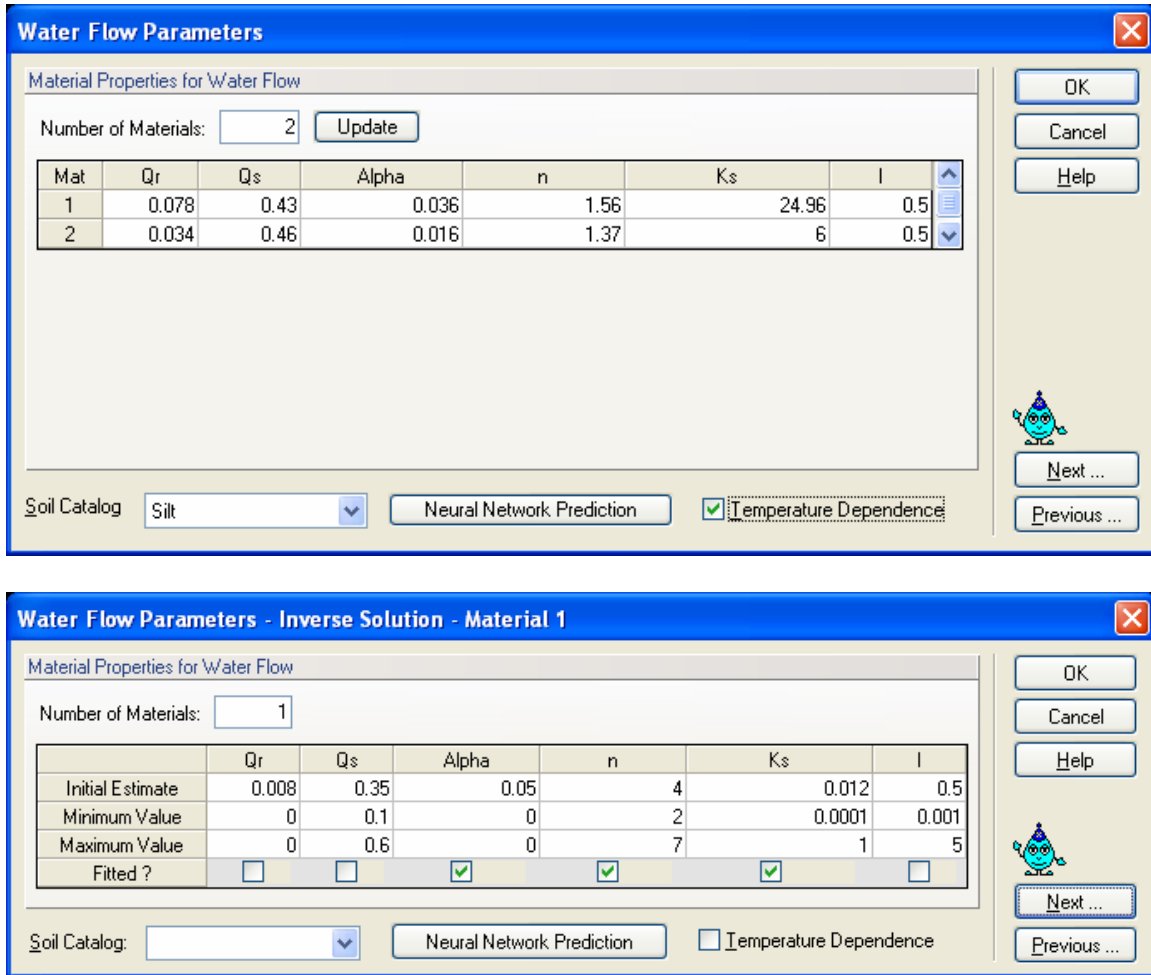
Figure 19. The Water Flow Parameters dialog window for direct (top) and inverse (bottom) problems.

Durner's [1994] model has three additional parameters: w_2 (w2), α_2 (Alpha2) [L^{-1}], and n_2 (n2), where w_2 is the weighting factor for the second overlapping region, and α_2 and n_2 are empirical parameters for the second region.

The hysteretic model has also three additional parameters: θ_{sw} (QsW) – the saturated water content of the main wetting branch, α_w (AlphaW) [L^{-1}] – the shape parameter of the main wetting branch, and K_{sw} (KsW) [LT^{-1}] - the saturated hydraulic conductivity associated with the main wetting branch (in case hysteresis also occurs in the conductivity function).

Temperature Dependence: Check this box if the hydraulic properties are considered to be temperature dependent. Using capillary theory, the influence of temperature on the soil water pressure head is then quantitatively predicted from the influence of temperature on surface tension, while the influence of temperature on the hydraulic conductivity is predicted from the influence of temperature on viscosity and the density of water.

Soil Catalog: The hydraulic parameters of selected soils were included into a catalog from which users can make selections. The parameters were taken from *Carsel and Parrish* [1988] (Table 7). Some caution is needed when using these parameter values since they only represent very approximate averages for different textural classes.

The following soil textural classes are represented in the soil hydraulic catalog:
        Sand
        Loamy Sand
        Sandy Loam
        Loam
        Silt Loam
        Sandy Clay Loam
        Clay Loam
        Silty Clay Loam
        Sandy Clay
        Silty Clay
        Clay

Neural Network Prediction: The program uses pedotransfer functions (PTFs) based on neural networks to predict *van Genuchten*'s [1980] water retention parameters and the saturated hydraulic conductivity (K_s) based on textural information (see Section 3.8 below).

When the parameter estimation option is selected, then users have to provide initial estimates of the optimized soil hydraulic parameters, specify which parameters are to be optimized (check appropriate checkboxes), and provide parameter constraints for the optimization. Entering zeros (the default values) for the minimum and maximum values signifies that the parameters are unconstrained.

Table 7. Soil hydraulic parameters for the analytical functions of *van Genuchten* [1980] for twelve textural classes of the USDA soil textural triangle according to *Carsel and Parish* [1988].

Textural class	θ_r [L^{3}L^{-3}]	θ_s [L^{3}L^{-3}]	α [cm^{-1}]	n [-]	K_s [cm d^{-1}]
Sand	0.045	0.430	0.145	2.68	712.8
Loamy Sand	0.057	0.410	0.124	2.28	350.2
Sandy Loam	0.065	0.410	0.075	1.89	106.1
Loam	0.078	0.430	0.036	1.56	24.96
Silt	0.034	0.460	0.016	1.37	6.00
Silty Loam	0.067	0.450	0.020	1.41	10.80
Sandy Clay Loam	0.100	0.390	0.059	1.48	31.44
Clay Loam	0.095	0.410	0.019	1.31	6.24
Silty Clay Loam	0.089	0.430	0.010	1.23	1.68
Sandy Clay	0.100	0.380	0.027	1.23	2.88
Silty Clay	0.070	0.360	0.005	1.09	0.48
Clay	0.068	0.380	0.008	1.09	4.80

Table 8. Soil hydraulic parameters for the analytical functions of *van Genuchten* [1980] for twelve textural classes of the USDA textural triangle as obtained with the Rosetta Lite program [*Schaap et al.*, 2001].

Textural class	θ_r [L^{3}L^{-3}]	θ_s [L^{3}L^{-3}]	α [cm^{-1}]	n [-]	K_s [cm d^{-1}]
Sand	0.053	0.375	0.035	3.18	643.
Loamy Sand	0.049	0.390	0.035	1.75	105.
Sandy Loam	0.039	0.387	0.027	1.45	38.2
Loam	0.061	0.399	0.011	1.47	12.0
Silt	0.050	0.489	0.007	1.68	43.7
Silty Loam	0.065	0.439	0.005	1.66	18.3
Sandy Clay Loam	0.063	0.384	0.021	1.33	13.2
Clay Loam	0.079	0.442	0.016	1.41	8.18
Silty Clay Loam	0.090	0.482	0.008	1.52	11.1
Sandy Clay	0.117	0.385	0.033	1.21	11.4
Silty Clay	0.111	0.481	0.016	1.32	9.61
Clay	0.098	0.459	0.015	1.25	14.8

3.8. *Neural Network Predictions*

The HYDRUS code was coupled with the Rosetta Lite DLL (Dynamically Linked Library) (Fig. 20), which was independently developed by Marcel Schaap at the U.S. Salinity Laboratory (mschaap@ussl.ars.usda.gov) [*Schaap et al*., 2001]. Rosetta implements pedotransfer functions (PTFs) which predict *van Genuchten*'s [1980] water retention parameters and the saturated hydraulic conductivity (K_s) in a hierarchical manner from soil textural class, the soil textural distribution, bulk density and one or two water retention points as input. Rosetta has its own help features containing all relevant information and references. Rosetta provides soil hydraulic parameters for the analytical functions of *van Genuchten* [1980] for twelve textural classes of the USDA textural triangle (Table 8).
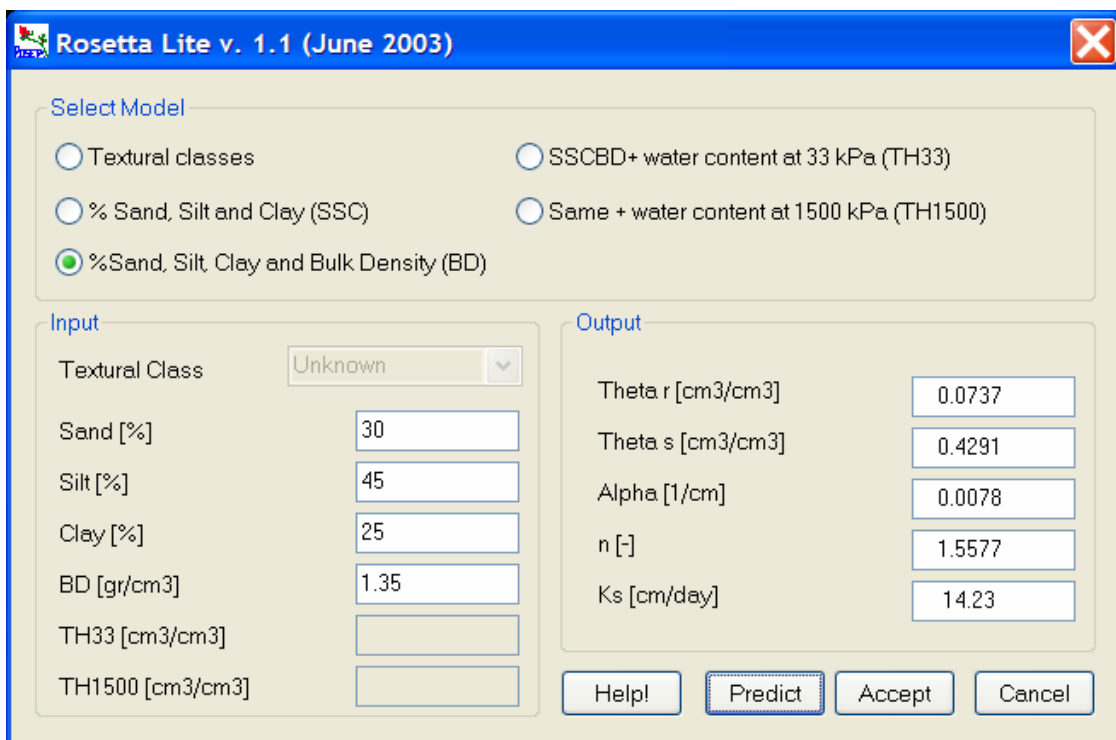
Figure 20. The Rosetta Lite (Neural Network Predictions) dialog window.

3.9. *Anisotropy in the Hydraulic Conductivity*

For two-dimensional problems users may need to specify the principal components of the anisotropy tensor, K_1^A and K_2^A, together with the angle ω_a between the principal direction of K_1^A and the *x*-axis of the global coordinate system for each element. This has been simplified for three-dimensional problems where user can specify one or more **Tensors of Anisotropy** (Fig. 21), which may be assigned to different parts of the transport domain. The anisotropy tensor is defined by three principal components, K_1^A (ConAX), K_2^A (ConAY), and K_3^A (ConAZ), and six coefficients a_{ij} that represent the cosine of angles between the *i*th principal direction of the tensor $\mathbf{K}^A$ and the *j*-axis of the global coordinate system (i.e., Cos(X-x), Cos(Y-y), Cos(Z-y), Cos(X-y), Cos(X-z), Cos(Y-z)).

Aniz	ConAX	ConAY	ConAZ	Cos(X-x)	Cos(Y-y)	Cos(Z-z)	Cos(X-y)	Cos(X-z)	Cos(Y-z)
1	1	1	1	1	1	1	0	0	0
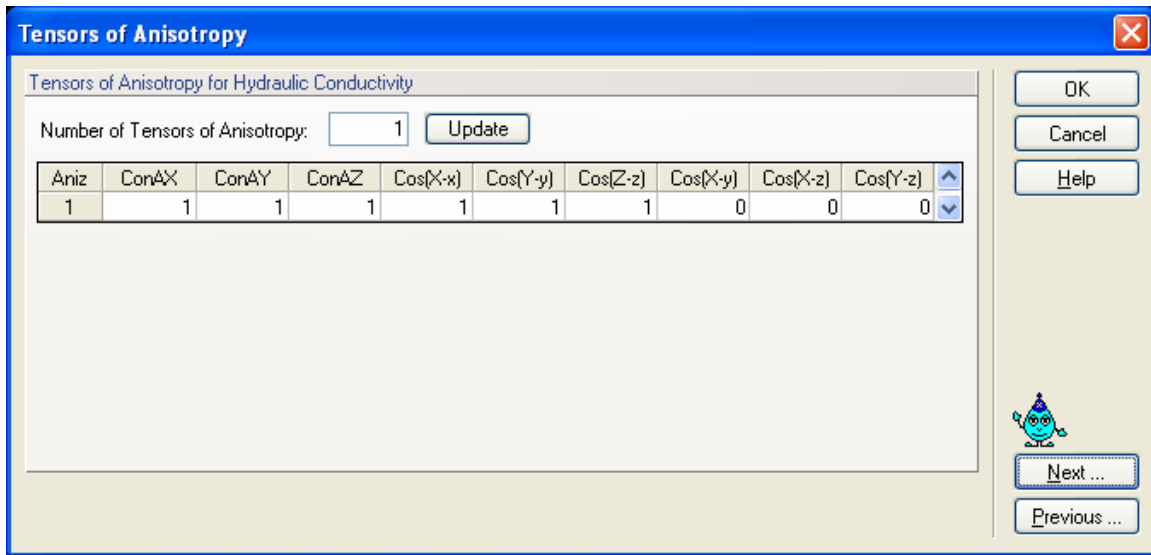
Figure 21. The Tensors of the Anisotropy dialog window.

3.10. *Solute Transport*

Basic information needed for defining solute transport problem are entered in the **Solute Transport** dialog window (Fig. 22). In this window users specify the **Space** and **Time Weighting Scheme**s, the **Iteration Criteria** (for nonlinear problems), and additional **Solute Information** such as mass units, pulse duration (if applicable), and number of solutes.
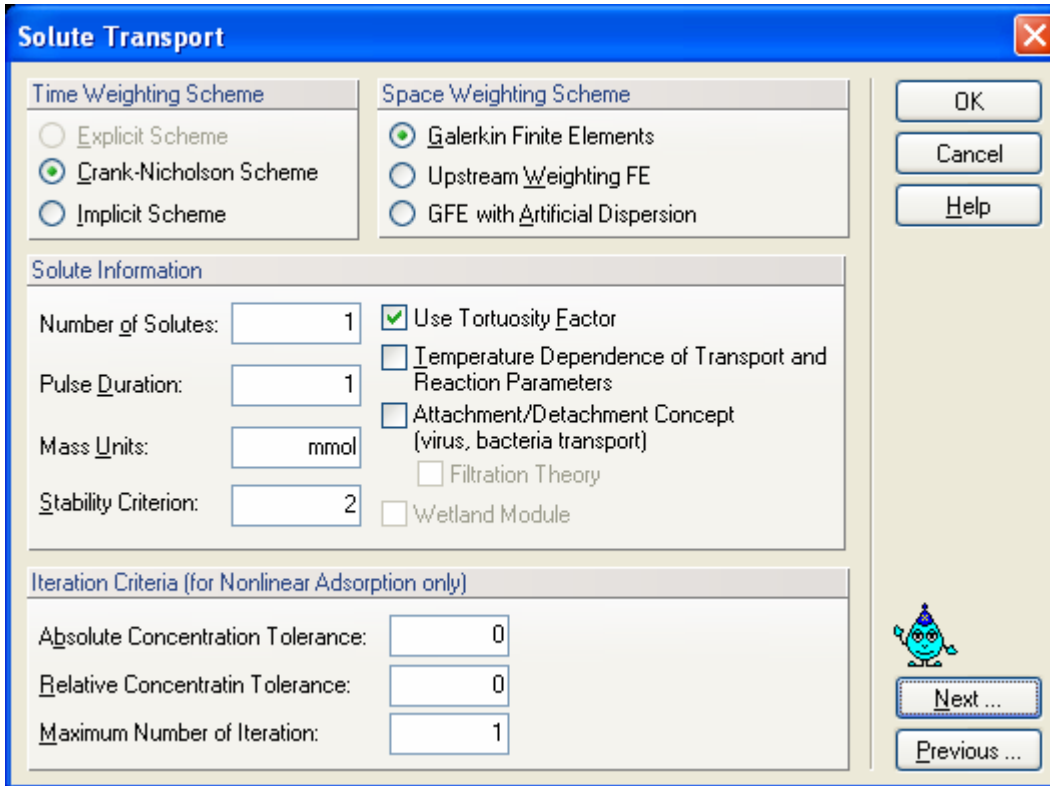
Figure 22. The Solute Transport dialog window.

a) Time Weighting Scheme

The **Time Weighting Scheme** defines the temporal weighing coefficient, ε, used in the numerical solution of the transport equation. The temporal weighting coefficient is equal to 0.0 for an *explicit scheme*, 0.5 for a *Crank-Nicholson time-centered implicit scheme*, and 1.0 for a *fully implicit scheme*. The structure of the final set of linear equations [G] {c} = {g}, obtained after the spatial and temporal discretization of the governing advection-dispersion equation, depends upon the value of the temporal weighing factor, ε. The explicit (ε=0) and fully implicit (ε=1) schemes require that the global matrix [G] and the vector {g} be evaluated at only one time level (the previous or current time level). The other two schemes require evaluation at both time levels. Also, the Crank-Nicholson and implicit schemes lead to an asymmetric banded matrix [G]. By contrast, the explicit scheme (ε=0) leads to a diagonal matrix [G] which is much easier to solve (but generally requires much smaller time steps).

The Crank-Nicholson centered scheme is recommended in view of solution precision. The fully implicit scheme also leads to numerical dispersion, but is better in avoiding numerical instabilities. The explicit scheme is most prone to numerical instabilities with undesired oscillations (and is currently disabled).

b) Space Weighting Scheme

HYDRUS provides three options for the **Space Weighting Scheme**, i.e., the regular *Galerkin Finite Elements* formulation, the *Upstream Weighting Finite Elements* formulation, and the *Galerkin Finite Elements* formulation *with Artificial Dispersion*.

While the **Galerkin Finite Elements** formulation is recommended in view of solution precision, **Upstream Weighting** is provided as an option in HYDRUS to minimize some of the problems with numerical oscillations when relatively steep concentration fronts are being simulated. For this purpose the second (flux) term of advective-dispersive equation is not weighted by regular linear basis functions, but instead using nonlinear functions [*Yeh and Tripathi*, 1990]. The weighing functions ensure that relatively more weight is placed on flow velocities of nodes located at the upstream side of an element.

Additional **Artificial Dispersion** may be added also to stabilize the numerical solution and to limit or avoid undesired oscillations in the Galerkin finite element results. Artificial dispersion is added such that a **Stability Criterion** involving Pe.Cr (the product of the Peclet number and the Curant number) [*Perrochet and Berod*, 1993] is satisfied. The recommended value for Pe.Cr is 2.0.

c) Solute Information

Number of Solutes	Number of solutes to be simulated simultaneously or involved in a decay chain reaction.
Pulse Duration	Time duration of the concentration pulse. Concentrations (flux or resident) along all boundaries, for which no time-variable boundary conditions are specified, are then set equal to zero for times larger than the "Pulse Duration".
Mass Units	Units to be printed to the output files or displayed in various graphs. Mass units have no effect on the calculations.
Stability Criterion	Product of the dimensionless Peclet and Curant numbers (Pe.Cr). This criterion is used either to add artificial dispersion in the *Galerkin Finite Elements with Artificial Dispersion* scheme or to limit the time step (leading to lower Courant numbers for a given Peclet number) for the *Galerkin Finite Elements* scheme.
Use Tortuosity Factor	Check this box when molecular diffusion coefficients in the water and gas phases are to be multiplied by a tortuosity factor according to the formulation of *Millington and Quirk* [1991].

Temperature Dependence	Check this box if the solute transport and reaction parameters are assumed to be temperature dependent.
Attachment/Detachment	Check this box if the solute is assumed to be subject to attachment/detachment to/from the solid phase. This process is often used in simulations of the transport of viruses, colloids, or bacteria.
Filtration Theory	Check this box if the attachment coefficient is to be calculated from filtration theory.
Wetland Module	Check this box if the Wetland module is to be used. This module is described in detail by *Langergraber and Šimůnek* [2005]. The Wetland module (for two-dimensional problems only) was developed to model biochemical transformation and degradation processes in subsurface flow constructed wetlands. The module considers the following components: dissolved oxygen, organic matter (3 fractions of chemical oxygen demand - COD: readily- and slowly-biodegradable, and inert), nitrogen compounds (ammonia, nitrite, nitrate, and dinitrogen), inorganic phosphorus, and heterotrophic and autotrophic microorganisms. Organic nitrogen and organic phosphorus are modeled as part of the COD. Heterotrophic bacteria are assumed to be responsible for hydrolysis, mineralization of organic matter (aerobic growth) and denitrification (anoxic growth), while Autotrophic bacteria are responsible for nitrification.

d) Iteration Criteria

The advection-dispersion solute transport equation becomes nonlinear when nonlinear adsorption is considered. Similarly as for the Richards equation, an iterative process must then be used to obtain solutions of the global matrix equation at each new time step. During each iteration a system of linearized algebraic equations is derived and solved using either Gaussian elimination or the conjugate gradient method. After inversion, the coefficients are re-evaluated using the initial solution, and the new equations are again solved. This iterative process continues until a satisfactory degree of convergence is obtained, i.e., until at all nodes the absolute change in concentration between two successive iterations becomes less than some concentration tolerance (defined in HYDRUS as the sum of an **Absolute Concentration Tolerance** and the product of the concentration and a **Relative Concentration Tolerance** (the recommended and default value is 0.001)). The **Maximum Number of Iterations** allowed during a certain time step needs to be specified (recommended value is 10). When the **Maximum Number of Iterations** is reached then the numerical solution is either (a) terminated for problems involving transient water flow or (b) restarted with a reduced time step for steady-state flow problems.

3.11. *Solute Transport Parameters*

Soil and *Solute Specific Transport Parameters* are specified in the **Solute Transport Parameters** dialog window (Fig. 23).

Figure 23. The Solute Transport Parameters dialog window.

The following **Soil Specific Parameters** (left part of the dialog window) are specified for each soil material:

Bulk.d.	Bulk density, ρ [ML^{-3}]
Disp.L.	Longitudinal dispersivity, D_L [L]
Disp.T.	Transverse dispersivity, D_T [L]
Frac	Dimensionless fraction of adsorption sites classified as type-1 sites, i.e., sites with instantaneous sorption when the **chemical nonequilibrium** option is considered [-]. Set this parameter equal to 1 when equilibrium transport is considered. *Frac* becomes the dimensionless fraction of adsorption sites in contact with mobile water when the **physical nonequilibrium** option is considered [-]. In that case *Frac* should be set equal to 1 when all sorption sites are in contact with mobile water.
ThImob	The immobile water content. Set equal to 0 when the **physical nonequilibrium** option is not considered.

The following **Solute Specific Parameters** (right part of the dialog window) are specified for each solute:

Diffus.W	Molecular diffusion coefficient in free water, D_w [L^{2}T^{-1}]
Diffus.G	Molecular diffusion coefficient in soil air, D_a [L^{2}T^{-1}]

3.12. *Solute Reaction Parameters*

The *Solute Reaction Parameters* and concentrations for *Boundary Conditions* are specified in the **Solute Reaction Parameters** dialog window (Fig. 24). Each solute has its own Solute Reaction Parameters dialog window.
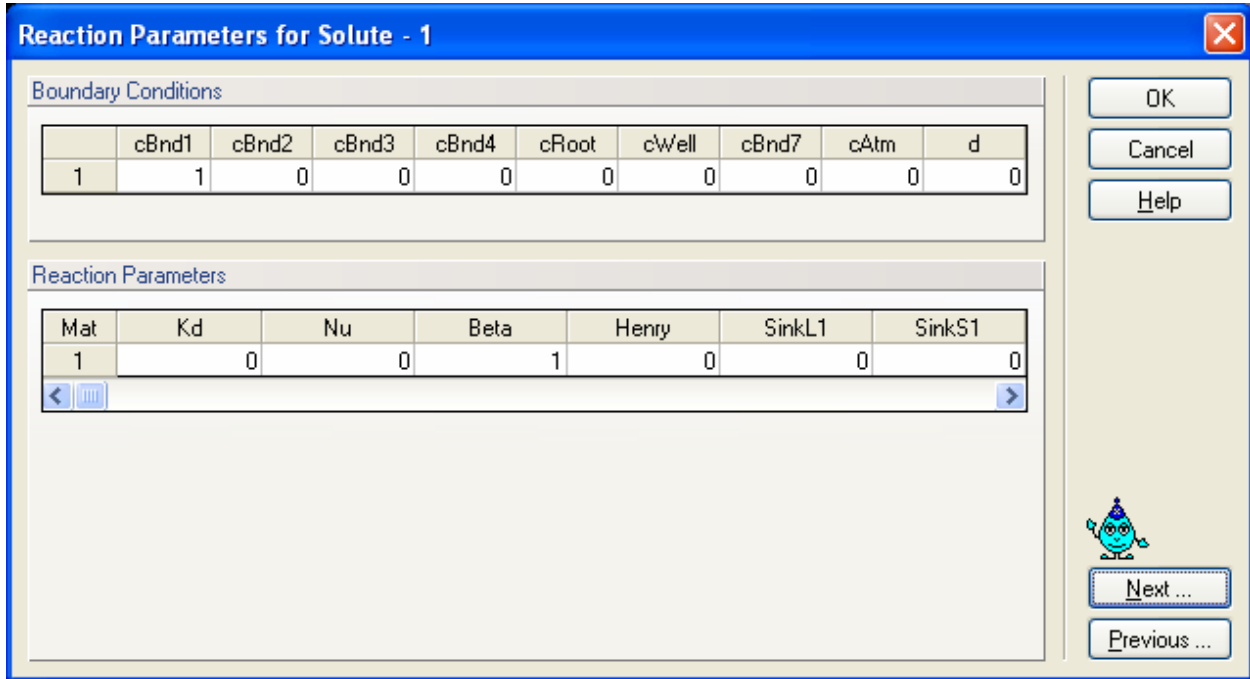
Figure 24. The Solute Reaction Parameters dialog window.

The following **Solute Reaction Parameters** are specified for each soil material:

Kd   Adsorption isotherm coefficient, k_s [$M^{-1}L^3$]
Nu   Adsorption isotherm coefficient, η [$M^{-1}L^3$]
Beta   Adsorption isotherm exponent, β [-]
Henry   Equilibrium distribution constant between liquid and gaseous phases, k_g [-]
SinkL1   First-order rate constant for dissolved phase, μ_w [T^{-1}]
SinkS1   First-order rate constant for solid phase, μ_s [T^{-1}]
SinkG1   First-order rate constant for gas phase, μ_g [T^{-1}]
SinkL1'   First-order rate constant for dissolved phase, μ_w' [T^{-1}], as part of a solute decay chain
SinkS1'   First-order rate constant for solid phase, μ_s' [T^{-1}], as part of a solute decay chain
SinkG1'   First-order rate constant for gas phase, μ_g' [T^{-1}], as part of a solute decay chain
SinkW0   Zero-order rate constant for dissolved phase, γ_w [$ML^{-3}T^{-1}$]
SinkS0   Zero-order rate constant for solid phase, γ_w [T^{-1}]
SinkG0   Zero-order rate constant for gas phase, γ_s [$ML^{-3}T^{-1}$]

Alpha   First-order rate coefficient for one-site or two-site nonequilibrium adsorption, mass transfer coefficient for solute exchange between mobile and immobile liquid regions, ω [T^{-1}]

When the **Attachment/Detachment Model** (Fig. 22) is used, then some parameters listed above are replaced with different parameters needed for the attachment/detachment model:

D_Soil   Diameter of the sand grains, d_c [L].
iPsi2   Type of blocking on the second sorption sites:
                  = 0: No blocking.
                  = 1: Langmuirian dynamics.
                  = 2: Ripening.
                  = 3: random sequential adsorption model.
                  = 4: depth dependent blocking coefficient.
iPsi1   Same for the first sorption sites.
SMax2   Parameter in the blocking function for the second sorption sites (s_{max} for blocking options (1), (2) and (3), and b for (4)).
AttachSolid2 First-order deposition (attachment) coefficient, k_a [T^{-1}], for the second sorption sites.
DetachSolid2 First-order entrainment (detachment) coefficient, k_d [T^{-1}], for the second sorption sites.
SMax1   Parameter in the blocking function for the first sorption sites.
AttachSolid1 First-order deposition (attachment) coefficient, k_a [T^{-1}], for the first sorption sites.
DetachSolid1 First-order entrainment (detachment) coefficient, k_d [T^{-1}], for the first sorption sites.

When **Filtration Theory** (Fig. 22) is used to calculate the attachment coefficient, then the following parameters must be entered instead:

D_Soil   Diameter of the sand grains, d_c [L].
D_Virus   Diameter of the particle, d_p (e.g., virus, bacteria) (e.g., = 0.95 μm or 0.95e-6 m) [L].
SMax2   Parameter in the blocking function for the second sorption sites (s_{max} for model (1)).
Stick. Eff2   Sticking efficiency, a [-], for the second sorption sites.
DetachSolid2 First-order entrainment (detachment) coefficient, k_d [T^{-1}], for the second sorption sites.
SMax1   Parameter in the blocking function for the first sorption sites.
Stick. Eff1   Sticking efficiency, a [-], for the first sorption sites.
DetachSolid1 First-order entrainment (detachment) coefficient, k_d [T^{-1}], for the first sorption sites.

Boundary Conditions:
Concentrations for time-independent Boundary Conditions are also specified in this dialog window.

cBnd1   Value of the concentration for the first time-independent boundary condition [ML^{-3}]. Set equal to zero if no time-independent boundary condition is specified. The same for cBnd2 through cBnd4

cRoot   Value of the concentration for the fifth time-independent boundary condition [ML^{-3}]. If water uptake is considered then *cRoot* is automatically used for the maximum concentration of water removed from the flow region by root water uptake. When zero is specified, then all solute is left behind in the soil and only a solute-free solution is being taken up. When the concentration is lower than *cRoot*, all solute is taken up. When the concentration is higher than *cRoot*, the excess solute stays behind. Set equal to zero if no fifth time-independent boundary condition and no solute uptake by roots is considered.

cWell   Value of the concentration for the sixth time-independent boundary condition [ML^{-3}]. If internal sources are specified, then *cWell* is automatically used for the concentration of water injected into the flow region through internal sources. Set equal to zero if no sixth time-independent boundary condition and no internal sources are specified.

cBnd7   Concentration of the incoming fluid for a volatile type boundary condition at the soil surface [ML^{-3}]. Set equal to zero if no volatile boundary condition is specified.

cAtm   Concentration above the stagnant boundary layer, g_{atm} [ML^{-3}], for a volatile type boundary condition. Set equal to zero if no volatile boundary condition is being specified.

d   Thickness of the stagnant boundary layer, *d* [L], for a volatile type boundary condition. Set equal to zero if no volatile boundary condition is being specified.

When the parameter estimation option is selected, then users have to provide initial estimates of the optimized solute transport parameters, specify which parameters are to be optimized (select appropriate checkboxes), and provide parameter constraints for the optimization. Zero values for minimum and maximum values signify that the parameters are unconstrained. The **Solute Transport and Reaction Parameters** dialog window for the inverse problem is not further shown here.

3.13. *Temperature Dependence of Solute Reaction Parameters*

Several of the diffusion (D_w, D_g), zero-order production (γ_w, γ_s, γ_g), first-order degradation (μ_w', μ_s', μ_g', μ_w, μ_s, and μ_g), and adsorption (k_s, k_g, β, η, ω) coefficients may be strongly dependent upon temperature. HYDRUS assumes that this dependency can be expressed by the Arrhenius equation [*Stumm and Morgan*, 1981]. This equation can be expressed in the general form

$$a_T = a_r \exp\left[\frac{E_a(T^A - T_r^A)}{R_u T^A T_r^A}\right]$$

where a_r and a_T are values of the coefficient being considered at a reference absolute temperature T_r^A and absolute temperature T^A, respectively; R_u is the universal gas constant, and E_a [$ML^2T^{-2}M^{-1}$] is the activation energy of the particular reaction or process being modeled.

The activation energy characterizing the temperature dependence of the solute transport and reaction parameters is entered in the dialog window shown in Figure 25.

Temperature Dependent Solute Transport and Reaction Parameters

Parameters

	SinkWater1	SinkSolid1	SinkGas1	SinkWater1'	SinkSolid1'	SinkGas1'
1	51213	0	0	81171	0	0
2	51213	0	0	81171	0	0
3	51213	0	0	81171	0	0

OK Cancel Help Next... Previous...
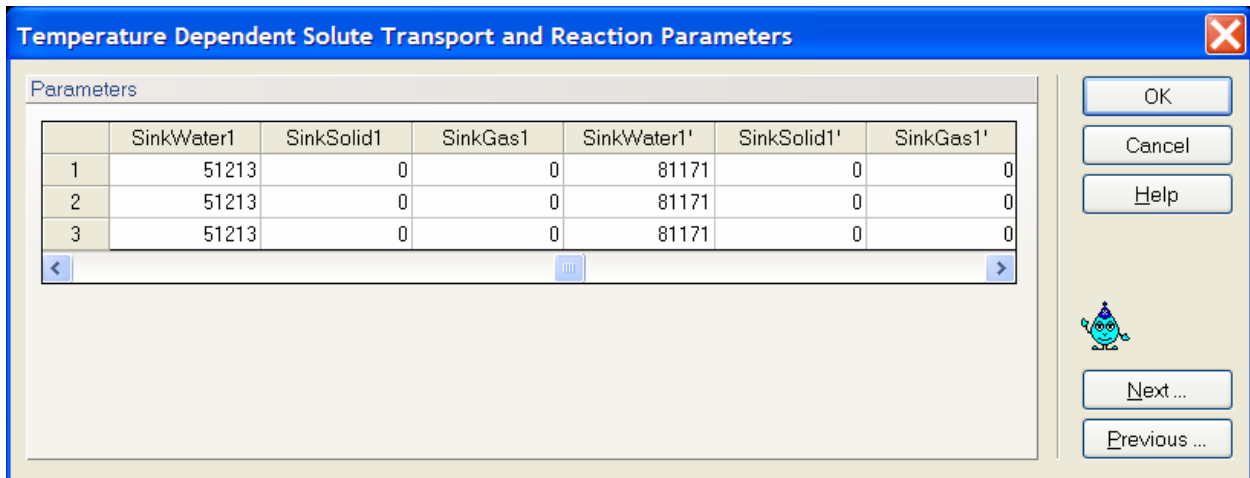
Figure 25. The Temperature Dependent Solute Transport and Reaction Parameters dialog window.

3.14. *Heat Transport Parameters*

Information needed for defining heat transport problem is entered in the **Heat Transport** dialog window (Fig. 26). In this window users specify *Heat Transport Parameters* and temperatures for the *Boundary Conditions*.
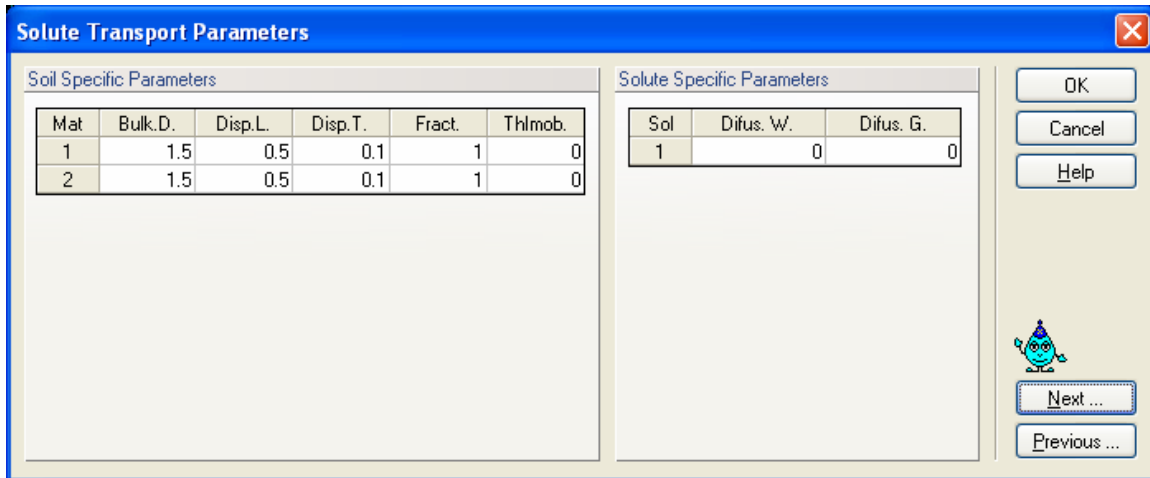

Figure 26. The Heat Transport Parameters dialog window.

The following **Heat Transport Parameters** (bottom part of the dialog window) are specified for each soil material:

Solid   Volume fraction of solid phase, θ_n [-]
Org.M.   Volume fraction of organic matter, θ_o [-]
Disp.L.   Longitudinal thermal dispersivity, λ_L [L]
Disp.T.   Longitudinal thermal dispersivity, λ_T [L]
b1   Coefficient b_1 in the expression for the thermal conductivity function [W/L/K]
b2   Coefficient b_2 in the expression for the thermal conductivity function [W/L/K]
b3   Coefficient b_3 in the expression for the thermal conductivity function [W/L/K]
Cn   Volumetric heat capacity of the solid phase, C_n [J/L^3/K]
Co   Volumetric heat capacity of organic matter, C_o [J/L^3/K]

Cw   Volumetric heat capacity of the liquid phase, C_w [J/L^3/K]

Boundary Conditions:
Temperatures for *Boundary Conditions* with time-independent boundary conditions are also specified in this dialog window.

TBound1   Value of the temperature for the first time-independent boundary condition [K]. Set equal to zero if no time-independent boundary condition is specified. The same for TBound2 through TBound4

TWell   Value of the temperature for the sixth time-independent boundary condition [K]. If internal sources are specified, then *TWell* is automatically used for the temperature of water injected into the flow region from sources in the transport domain. Set equal to zero if no sixth time-independent boundary condition and no internal sources are specified.

The boundary condition at the soil surface may be approximated using a sinus wave with the maximum one hour after noon and the minimum one hour after midnight as follows:

$$T = T_0 + A\cos\left[2\pi\left(\frac{t-13}{24}\right)\right]$$

where T_0 is the average temperature at the soil surface [K], A is the **Temperature Amplitude** at the soil surface [K], and p is the **Time Interval** for completion of one sine wave temperature (usually 1 day, the default value). The second part of the sine term is included to set the maximum temperature at 1 p.m.

Default values for the parameters in the **Thermal Conductivities** of three textural classes (sand, loam and clay) are provided by HYDRUS [*Chung and Horton*, 1987]. Default volumetric heat capacities for the solid phase, organic matter, and liquid phase are also given (*Set Default Volumetric Heat Capacities*).

When the parameter estimation option is selected, then users must provide initial estimates of the optimized heat transport parameters, specify which parameters are to be optimized (check appropriate checkboxes), and provide parameter constraints for the optimization. Zero values for the minimum and maximum values signify that parameters are unconstrained. The **Heat Transport Parameters** dialog window for the inverse problem is not further shown here.

3.15. *Root Water Uptake Model*

Users may select a particular **Water Uptake Reduction Model** and a **Solute Stress Model** in the **Root Water Uptake Model** dialog window (Fig. 27).
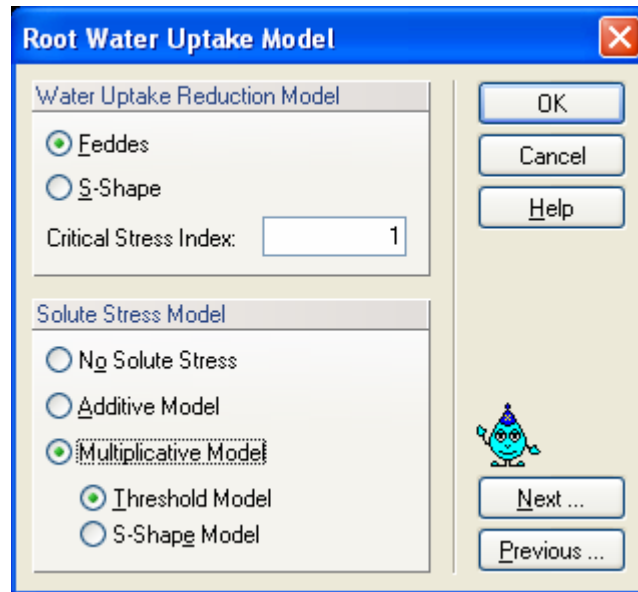
Figure 27. The Root Water Uptake Model dialog window.

Water Uptake Reduction Model
Either a water stress response function suggested by *Feddes* et al. [1978] or an S-shaped function suggested by *van Genuchten* [1985] can be used to reduce the potential root water uptake to the actual water uptake rate (Fig. 28). Root water uptake with compensation can be simulated when the *Critical Stress Index* is smaller than one (see the Technical Manual; *Šimůnek et al.* [2006]).

Solute Stress Model
The effect of salinity stress on the root water uptake can be either neglected (*No Solute Stress*) or considered using the *Additive* or *Multiplicative* models, i.e., salinity stress is either added to water stress, or uptake reduction due to water stress and salinity stress are multiplied. When the multiplicative model is used for salinity stress, one can use either the *Threshold Model* [*Maas*, 1990] or an *S-Shaped Model* [*van Genuchten*, 1985] (Fig. 29).

3.16. *Root Water Uptake Parameters*

Parameters for the water and salinity stress response functions are specified in the **Root Water Uptake Parameters** dialog window (Fig. 28 and Fig. 29, respectively).
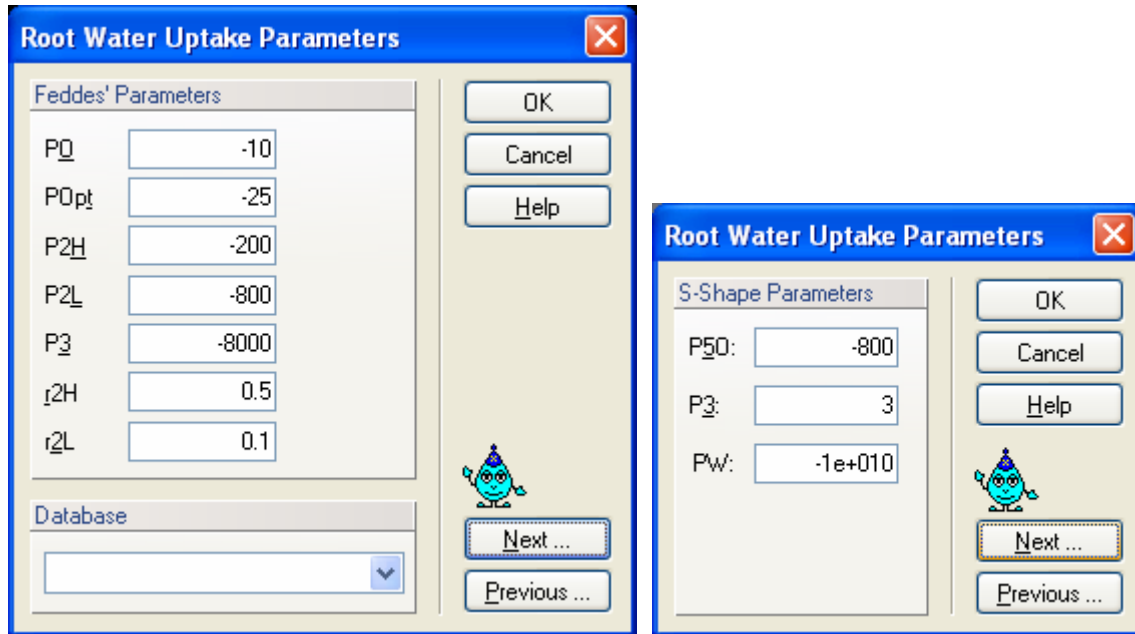
Figure 28. The Root Water Uptake Parameters dialog window for the water stress response function of *Feddes et al.* [1978] (left) and *van Genuchten* [1985] (right).

The **Root Water Uptake Parameters** for the water stress response function suggested by *Feddes et al.* [1978] (Fig. 28, left) are described in detail in the HYDRUS technical manual. Water uptake in this model is assumed to be zero close to saturation (i.e. wetter than some arbitrary "anaerobiosis point" **P0**). Root water uptake is also zero for pressure heads less (more negative) than the wilting point (**P3**). Water uptake is considered optimal between pressure heads **Popt** and **P2**, whereas for pressure heads between **P2** and **P3** (or **P0** and **Popt**) water uptake decreases (or increases) linearly with pressure head.

P0   Value of the pressure head [L] below which roots start to extract water from the soil.

POpt   Value of the pressure head [L] below which roots extract water at the maximum possible rate.

P2H   Value of the limiting pressure head [L] below which roots cannot longer extract water at the maximum rate (assuming a potential transpiration rate of *r2H*).

P2L   As P2H, but for a potential transpiration rate of *r2L*.

P3   Value of the pressure head [L] below which root water uptake ceases (usually taken at the wilting point).

r2H   Potential transpiration rate $[LT^{-1}]$ (currently set at 0.5 cm/day).

r2L   Potential transpiration rate [LT⁻¹] (currently set at 0.1 cm/day).

The above input parameters permit one to make the variable P2 a function of the potential transpiration rate, T_p (P2 presumably decreases at higher transpiration rates). HYDRUS currently implements the same linear interpolation scheme as used in several versions of the SWATRE code (e.g., *Wesseling and Brandyk*, 1985). The interpolation scheme is defined in the manual.

A database of suggested values for different plants for the *Feddes et al.* [1978] model is provided in HYDRUS based on studies by *Wesseling* [1991] and *Taylor and Ashcroft* [1972].

The **Root Water Uptake Parameters** for the S-shaped water stress response function as suggested by *van Genuchten* [1985] (Fig. 28, right) are as follows:

P50   The coefficient, *h50*, in the root water uptake response function associated with water stress [L]. Root water uptake at this pressure head is reduced by 50%.

P3   The exponent, *p3*, in the root water uptake response function associated with water stress [-]; its recommended and default value is 3.

We have additionally included a parameter **PW**, i.e., pressure head at the wilting point [L], below which transpiration stops.
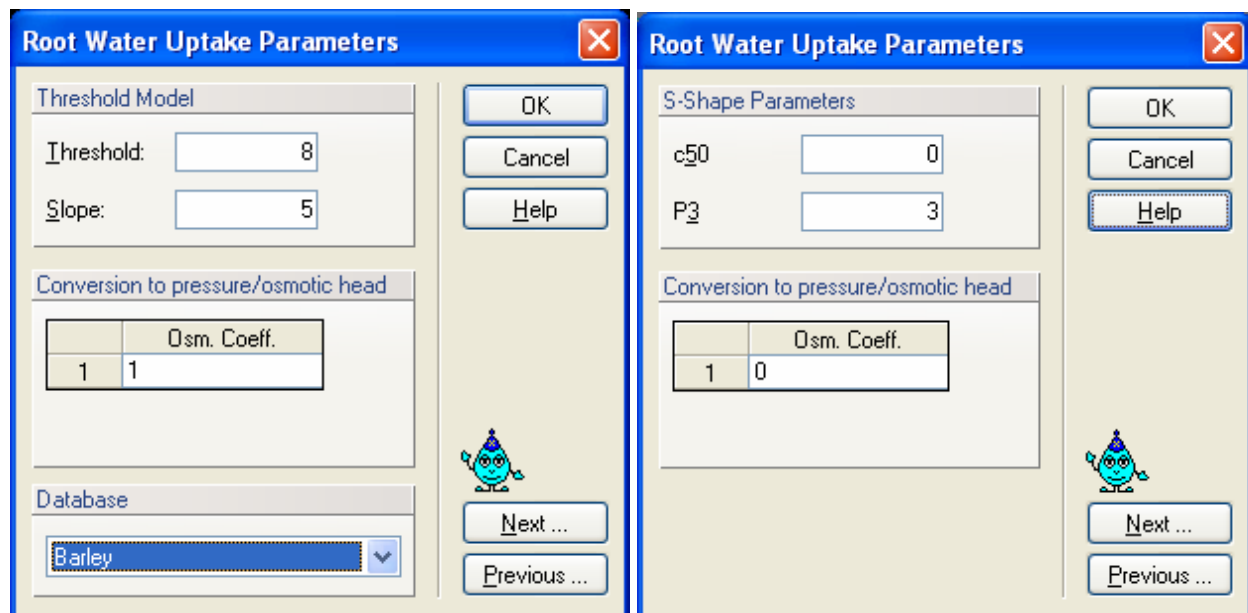
Figure 29. The Root Water Uptake Parameters dialog window for the solute stress response function based on the threshold model (left) and S-shaped model of *van Genuchten* [1985] (right).

The **Root Water Uptake Parameters** for the *Threshold Model* [*Maas*, 1990] of the salinity stress response function (multiplicative) (Fig. 29, left) are as follows:

Threshold Value of the minimum osmotic head [L] (the salinity threshold) above which root water uptake occurs without a reduction.

Slope   Slope of the curve determining the fractional root water uptake decline per unit increase in salinity below the threshold.

The **Root Water Uptake Parameters** for the *S-Shaped Model* [*van Genuchten*, 1985] of the salinity stress response function (multiplicative) (Fig. 29, right) are as follows:

P3   The exponent, *p*, in the root water uptake response function associated with salinity stress [-]. The recommended value is 3.

c50   The coefficient, *h50*, in the root water uptake response function associated with salinity stress [L]. Root water uptake at this osmotic head is reduced by 50%.

Both salinity stress response functions require a coefficient (*Osmotic Coefficient*) that transforms concentrations into equivalent osmotic pressure heads (Fig. 29).

3.17. *Root Distribution Parameters*

The spatial distribution of the roots can be specified using the **Root Distribution Parameters** dialog window (Fig. 30). The following two- and three-dimensional root distribution functions are implemented in HYDRUS [*Vrugt et al.*, 2001, 2002]:

$$b(x,z) = \left(1 - \frac{z}{Z_m}\right)\left(1 - \frac{x}{X_m}\right) e^{-\left(\frac{p_z}{Z_m}\left|z^* - z\right| + \frac{p_r}{X_m}\left|x^* - x\right|\right)}$$

$$b(x,y,z) = \left(1 - \frac{x}{X_m}\right)\left(1 - \frac{y}{Y_m}\right)\left(1 - \frac{z}{Z_m}\right) e^{-\left(\frac{p_x}{X_m}\left|x^* - x\right| + \frac{p_y}{Y_m}\left|y^* - y\right| + \frac{p_z}{Z_m}\left|z^* - z\right|\right)}$$

where X_m, Y_m, and Z_m are the maximum rooting lengths in the x-, y-, and z- directions [L], respectively; x, y, and z are distances from the origin of the plant (tree) in the x-, y-, and z-directions [L], respectively; p_x [-], p_y [-], p_z [-], x^* [L], y^* [L], and z^* [L] are empirical parameters (in Fig. 30 indicated as *Depth of Maximum Intensity* or *Radius of Maximum Intensity*, and $b(x,z)$ and $b(x,y,z)$ denote two- and three-dimensional spatial distribution of the potential root water uptake [-]. See *Vrugt et al.* [2001, 2002] for different configurations of the normalized spatial distribution of potential root water uptake rate.
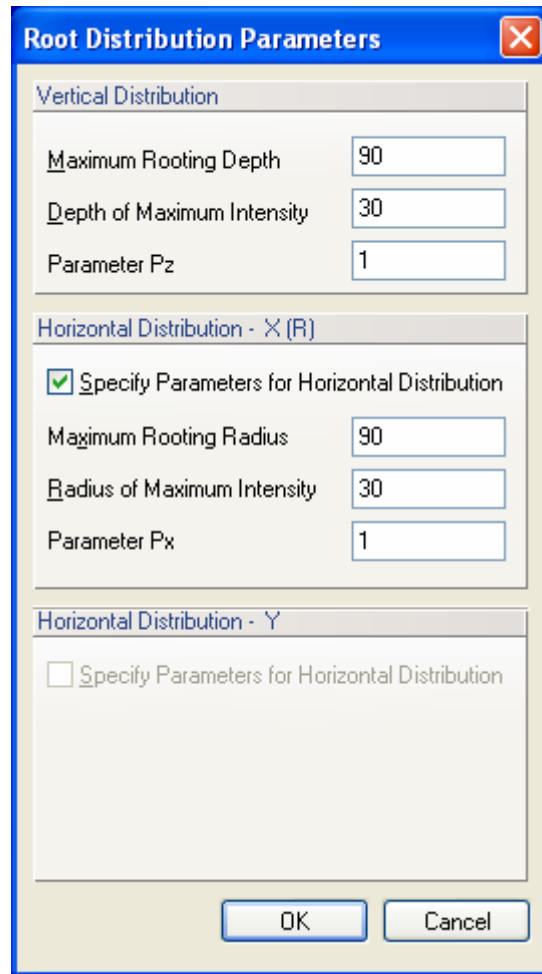
Figure 30. The Root Distribution Parameters dialog window.

3.18. *Time Variable Boundary Conditions*

The **Time Variable Boundary Conditions** dialog window is shown in Figure 31.
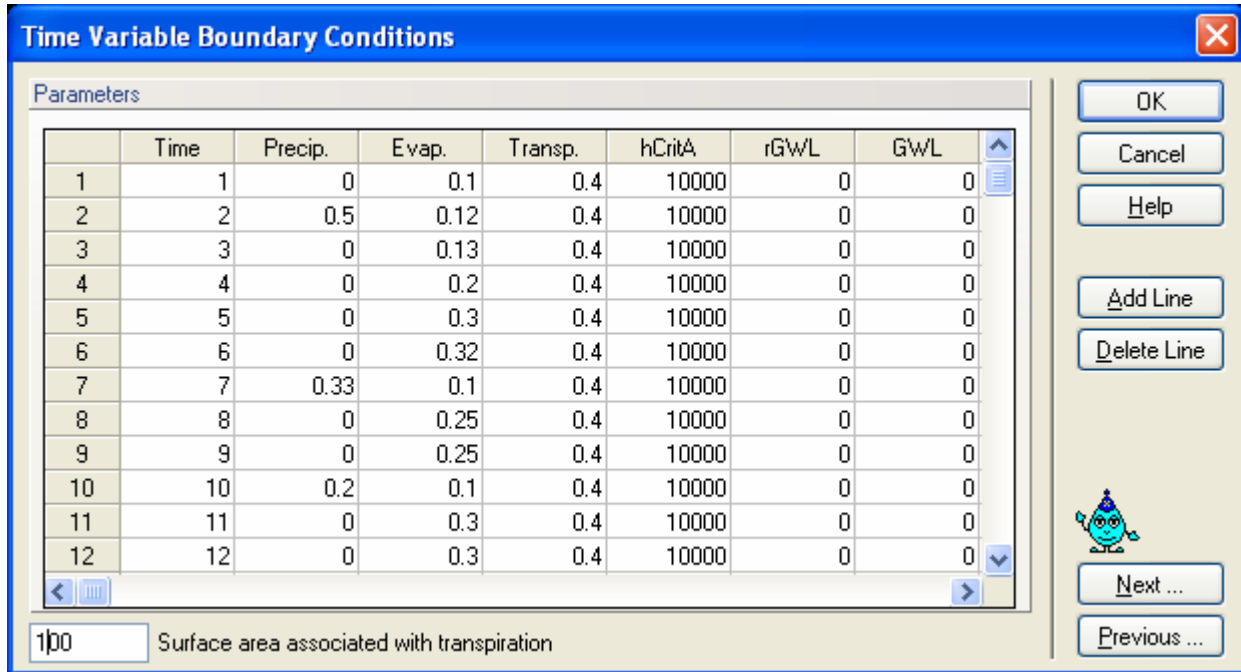
Figure 31. The Time Variable Boundary Conditions dialog window.

The following variables are specified in the **Time Variable Boundary Conditions** dialog window:

Time Time for which a data record is provided [T].

Precip Precipitation rate $[LT^{-1}]$ (in absolute value) (applied to the atmospheric boundary).

Evap Potential evaporation rate $[LT^{-1}]$ (in absolute value) (applied to the atmospheric boundary).

Trans Potential transpiration rate $[LT^{-1}]$ (in absolute value).

hCritA Absolute value of the minimum allowed pressure head at the soil surface [L] (applied to the atmospheric boundary).

rGWL1 Drainage flux $[LT^{-1}]$ across the bottom boundary, or another time-dependent prescribed flux boundary condition (positive when water leaves the flow region); set to zero when no time-dependent flux boundary condition is specified. Same for rGWL2, rGWL3, or rGWL4.

GWL1 Groundwater level [L] (usually negative), or other time-dependent prescribed head boundary condition; set equal to zero when no time-dependent head boundary condition is specified. Same for GWL2, GWL3, or GWL4.

Temp1   The first time-dependent temperature [K] that can be used for nodes with time variable boundary conditions (atmospheric BC, variable head/flux BC) (is not specified when heat transport or time variable boundary conditions are not considered).

Temp2   The second time-dependent temperature [K] that can be used for nodes with time variable boundary conditions (is not specified when heat transport or time variable boundary conditions are not considered).

Conc1   The first time-dependent solute concentration [ML^{-3}] that can be used for nodes with prescribed time variable boundary conditions (atmospheric BC, variable head/flux BC) (not specified when solute transport is not considered).

Conc2   The second time-dependent solute concentration [ML^{-3}] that can be used for nodes with prescribed time variable boundary conditions (atmospheric BC, variable head/flux BC) (not specified when solute transport is not considered).

Conc3   The third time-dependent solute concentration [ML^{-3}] that can be used for nodes with prescribed time variable boundary conditions (atmospheric BC, variable head/flux BC) (not specified when solute transport is not considered).

The last three entries are entered for each solute.

The table in Figure 31 can be edited by manually adding or deleting lines. The table has a capacity for about 32,000 records (depends on the number of columns). When a longer time record is to be simulated, then one needs to directly edit the Atmosph.in input file in the working directory using any standard software, such as MS Excel. Data for the **Time Variable Boundary Conditions** can be prepared in any spreadsheet software and then copied into the table using Windows paste hot keys (i.e., Ctrl+V).

The total number of atmospheric data records is given in the Main Time Information dialog window (Fig. 15).

3.19. *Constructed Wetlands*

Parameters for constructed wetlands are entered in the **Constructed Wetland Model Parameters I and II** dialog windows (Fig. 32 and 33, respectively). This module is described in detail by *Langergraber and Šimůnek* [2005]. The Wetland module (two-dimensional) was developed to model biochemical transformation and degradation processes in subsurface flow constructed wetlands. The module considers the following components: dissolved oxygen, organic matter (3 fractions of chemical oxygen demand - COD: readily- and slowly-biodegradable, and inert), nitrogen compounds (ammonia, nitrite, nitrate, and dinitrogen), inorganic phosphorus, and heterotrophic and autotrophic micro-organisms. Organic nitrogen and organic phosphorus are modeled as part of the COD. Heterotrophic bacteria are assumed to be responsible for hydrolysis, mineralization of organic matter (aerobic growth) and denitrification (anoxic growth), while autotrophic bacteria are responsible for nitrification.

Figure 32. The Constructed Wetland Model Parameter I dialog window.

Figure 33. The Constructed Wetland Model Parameter II dialog window.

4. Geometry of the Transport Domain

The transport domain may be defined using relatively simple two-dimensional rectangular (Fig. 10) or three-dimensional hexahedral (Fig. 11) objects. In that case, the dimensions and other parameters of the transport domain are specified numerically using either the *Rectangular* (Fig. 10) or *Hexahedral Domain Definition* (Fig. 11) dialog windows. In both of these cases the transport domain is discretized into a structured finite element mesh. Alternatively, a more general geometry can be defined from basic boundary objects, such as points, lines, splines, polylines, arcs and/or circles. Boundary curves can consist of any combination of polylines, arcs, circles or cubical splines. The program permits one to specify internal boundaries (e.g., drains, wells, impermeable objects), as well as internal curves. A user can define from these boundary objects either a two-dimensional transport domain or the base plane (**base surface**) of the three-dimensional domain. In both cases the two-dimensional transport domain or the base plane of the three-dimensional domain is discretized into an unstructured finite element mesh.

4.1. *Boundary Objects*

The **computational domain** (the two-dimensional transport domain or the **base surface** of the three-dimensional domain) is formed by an arbitrary number of mutually nonintersecting **curves**. Each curve can be formed by connecting an arbitrary number of **objects**. Objects are defined by nodes, the positions of which can be specified either graphically with the mouse with possibilities to use **grid** alignment (Fig. 78) or by numerically defining their coordinates *X*, *Y*, and *Z*. It is also possible to read in the objects with a large number of nodes (**spline**, **polyline**) from a file. The order of inputting particular objects is arbitrary; the code automatically forms the desired curves. In order to have a physically realistic domain, only one closed outer curve can exists (for multicomponent domains such a curve must exist for each component of the domain). The domain can have an arbitrary number of **holes** or **internal curves**. The consistency of the geometry can be verified at any time using the command **Check Geometry** (**Tools Menu**). Any change in geometry can be undone using the **undo** command (up to ten levels backward in time) or redone using the **redo** command (again up to 10 levels).

An object type (e.g., polyline, spline, arc, or circle) must be selected first when designing a new object from the **Edit Bar** on the right side of the view window, or from menu (e.g., Insert->Domain Geometry->Nodes). Then points defining a particular object should be entered. The manner in which nodes are entered depends on the selected input style. When entering data graphically, an appropriate **Work Plane** (*x-y y-z*, *x-z*) and a grid with appropriately set parameters (Fig. 78 and Section 8.1.2) can be used to facilitate the input, while the coordinates *X*, *Y*, and *Z* of the cursor are continuously displayed in the bottom right corner of the window.

It is possible to edit existing objects by double-clicking on a particular object. The current selection, displayed in yellow, may be modified (edited) using the following operations: **delete**, **copy** (Fig. 38), **move** (Fig. 38), **rotate** (Fig. 39, left) and **mirror** (Fig. 39, right). Note that, in addition to objects, particular nodes of an object can belong to a selected set as well, in which case the edit operations are carried out also for these points. Editing of selected objects (e.g., moving objects) also depends on the currently selected input style. The objects are moved with

the cursor when the graphical mode is selected, while in numerical mode a vector of translation (*X, Y*, and *Z*) must be specified. It is possible to directly edit nodes of objects with the commands **insert point** (Fig. 34), **delete point**, and/or **move point**.

Before saving the data, an option is always displayed whether or not to verify the consistency of the geometry. We strongly recommend to regularly perform this test in order to prevent errors in subsequent calculations, e.g., during mesh generation.

Table 9. Definition of terms related to geometry design.

Surface
A **surface** (the **computational domain** of two-dimensional applications) is defined by a finite number of continuous disjunctive bounded two-dimensional subdomains. This means that the domain can be multicomponent, but that each of its components must have only one **outer boundary curve**. The domain can contain any finite number of **internal holes** or **internal curves**. The **base surface** is a two-dimensional **surface** that can be extended into a three-dimensional **solid** (the **computational domain** of three-dimensional applications) using **thickness vectors**.

Computational Domain
The term **computational domain** is used for a particular **base surface** in two-dimensional applications and a **solid** in three-dimensional applications. **Computation domain** can be formed using several surfaces that can touch each other, but can not overlap. It is possible to create an opening in the base surface and then enter another surface into it. Division of the transport domain into individual surfaces enables easier work with it, since the program creates for each surface its own section and users can then specify different domain properties and initial and boundary conditions on these sections.

Curves
A **curve** is a set consisting of a finite number of objects connected by boundary nodes. Except at point nodes, objects cannot intersect each other or themselves. A curve can be open or closed.

Outer Boundary Curves
An outer boundary curve is a boundary **curve** with the following properties: the curve is closed, positively oriented (i.e., in a counter-clockwise direction), does not intersect any other curve or itself, and has the computational domain (surface) located on its left side (in the sense of positive orientation), while the right side is not part of a computational domain.

Openings
An opening is defined by one or more boundary **curves** that form a closed boundary.

Internal Curves
An internal curve must be located entirely within the computational domain. It can touch (but not intersect) the outside boundary of the computational domain at its definition points. An internal curve can be open or closed, and can intersect itself, provided the intersect occurs at a definition point of the internal curve.

Objects
The following types of geometric objects can be used:
- points
- polylines
- arcs
- circles
- splines
- single points

Lines
Lines and polylines are the most commonly used objects for describing boundaries of a domain or internal curves. Lines are defined by two points specifying their beginning and end. Several lines can be connected to form "polylines" by connecting the beginning and end of two neighboring lines. Nodes can not coincide, while lines can not intersect each other.

Arcs
An arc can be defined either by a) **three points** on its circumference, b) a **center,** a **radius, two angles** (starting and final angle) and its orientation, and c) **two points**, a **center**, and a **radius**.

Circles
A circle can be defined either by **a center and a radius**, or by **three different points**.

Thickness Vectors
The term **thickness vector** is used for a vector perpendicular to the **base surface** that extends the Base Surface to form a **solid** (three-dimensional **computational domain**).

Solids
The term **solid** represents a three dimensional **computational domain** that is formed by the **base surface** and **thickness vectors**.

4.1.1. *Points*

Points can be either used to define **Boundary Objects** or can be located inside of the **Surface** (computational domain) and not be associated with any boundary object (**Single Points**). Points can be entered either graphically using a cursor (most common) or using the **New Point** dialog window (identical to the **General Tab** of the **New Point** dialog, Fig. 34). To enter a new point graphically, select the command *Insert->Domain Geometry->Point->Graphically* from the menu, or **Points** from the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar** at the right side of the **View Window**, and then enter the points using a cursor. Double-clicking on an existing point will recall the **Edit Point** dialog window (Fig. 34). In addition to the **General Tab** (Fig. 34, left) in the **New Point** dialog window, the **Edit Point** dialog window has also the **FE-Mesh Tab** (Fig. 34, right). Coordinates of a point and its number are entered in the **General Tab**, while the **FE-Mesh Refinement** at a given point can be defined in the **FE-Mesh Tab**.
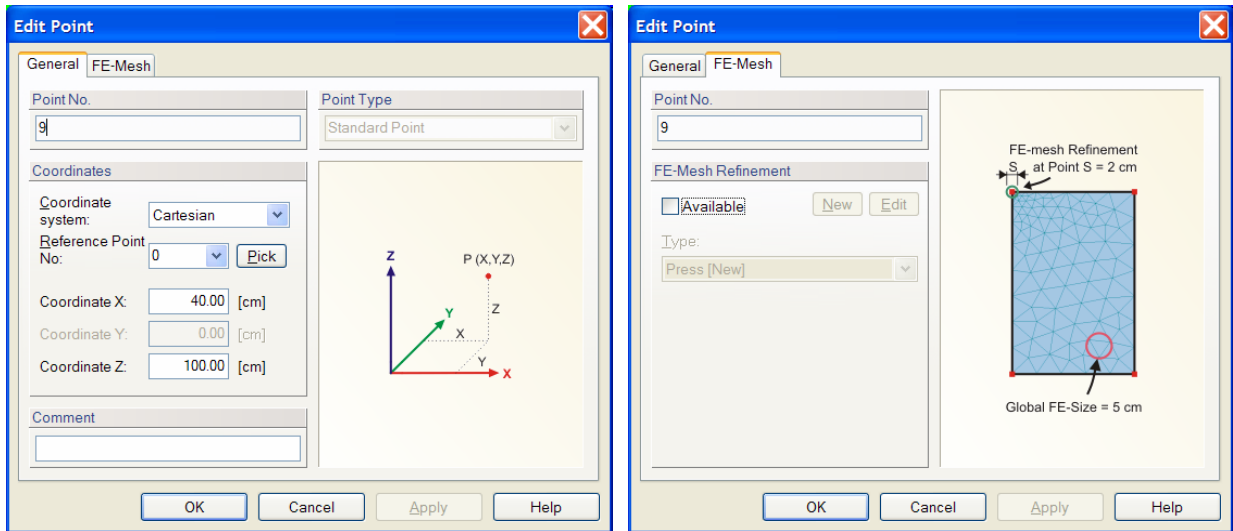
Figure 34. The Edit Point dialog window.

4.1.2. *Lines and Polylines*

Lines and **Polylines** are the most commonly used objects for describing the boundaries of a two-dimensional domain and its internal curves. Similarly as above for **Points**, **Lines** and **Polylines** can be entered either graphically using a cursor (most common) or using the **New Line** dialog window (identical to the **General Tab** of the **Edit Line** dialog, Fig. 35). When entering a new line graphically, users can select the command *Insert->Domain Geometry->Line(Polyline)->Graphically* from the menu, or **Line - Abscissa** or **Line - Polyline** from the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar** at the right side of the **View Window** and then enter the lines using a cursor. Double-clicking on an existing line will recall the **Edit Line** dialog window (Fig. 35). Before using the **New Line** dialog window, a user needs to first define points which are then used to define the line. In addition to the **General Tab** (Fig. 35, left) in the **New Line** dialog window, the **Edit Curve** dialog window has also the **FE-Mesh Tab** (Fig. 35, right). Points defining the line are entered in the **General Tab**, while the **FE-Mesh Refinement** along a given line can be defined in the **FE-Mesh Tab**.
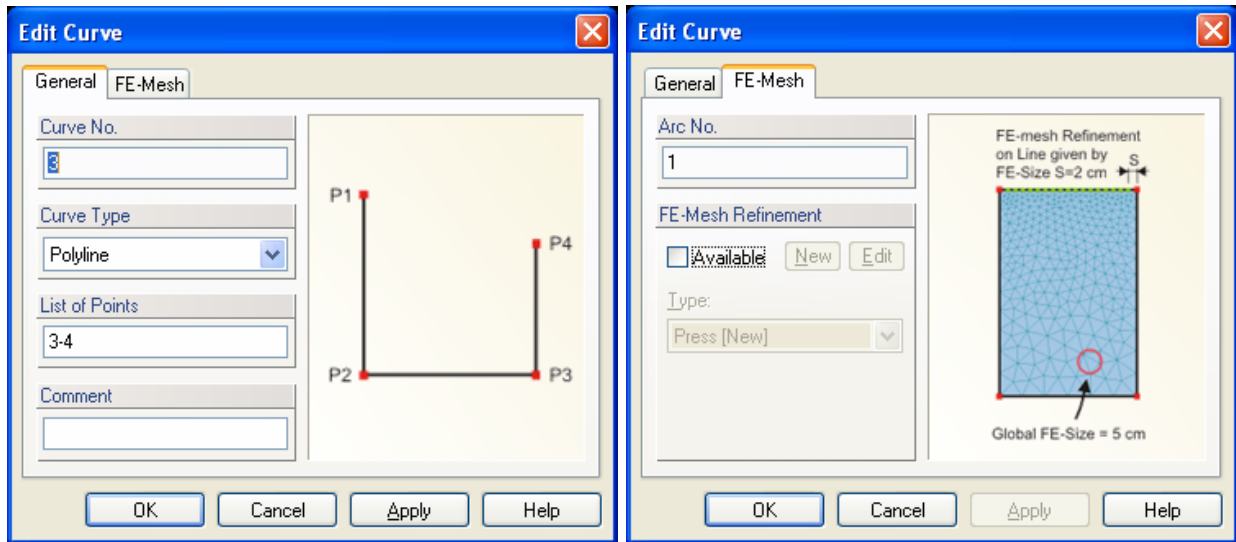
Figure 35. The Edit Curve dialog window.

4.1.3. *Arcs and Circles*

An **arc** is part of a **circle**. An **Arc** can be defined either by a) **three points** on its circumference, b) **a center, a radius, two angles** (starting and final angle) and its orientation, or c) by **two points**, a **center**, and a **radius**. Again, **arcs** can be entered either graphically using a cursor (most common) or using the **New Line** dialog window (Fig. 36). When entering a new **arc** or circle graphically, a user must select the command *Insert->Domain Geometry->Arc->Graphically* from the menu, or one of the following commands (a) **Arc by 3 Points**, b) **Arc by 2 Points and R**, or c) **Arc by Center, R and Angle**) from the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar** at the right side of the **View Window** and then enters **arc** using a cursor.

In addition to the **General Tab** (Fig. 36, left) in the **New Line** dialog window, the **Edit Line** dialog window has also the **Arc Tab** (Fig. 36, right). A list of points defining the arc are given in the **General Tab**, while coordinates of points defining the arc, its center and other parameters are entered in the **Arc Tab**. Double-clicking on an existing line will recall the **Edit Curve** dialog window (Fig. 36). The **Edit Curve** dialog window (Fig. 36) has an additional **FE-Mesh Tab** (similarly as for a line in Fig. 35; not shown here) where a user can refine the **FE-Mesh** along a given arc.

72
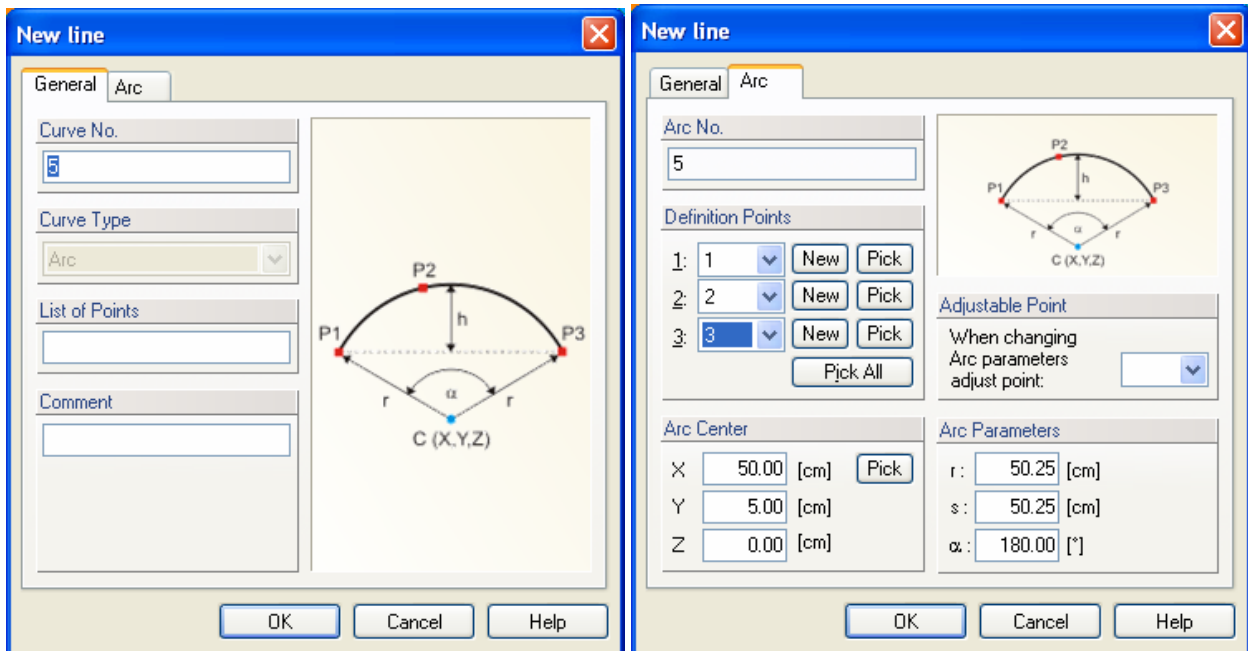
Figure 36. The New Line (Arc) dialog window.

A **circle** can be defined either by **a center and a radius**, or by **three different points**. Again, **circles** can be entered either graphically using the cursor (most common) or using the **New Line** dialog window (Fig. 36). A new **circle** can be entered graphically, by selecting the command *Insert->Domain Geometry->Arc(Circle)->Graphically* from the menu, or by using one of the following commands (a) **Circle by 3 Points**, or b) **Circle by Center and Radius**) from the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar** at the right side of the **View Window**, and then entering lines using the cursor. The list of points defining the circle must be entered in the **General Tab** (Fig. 37, left), while coordinates of points defining the circle, its center and radius are entered in the **Circle Tab** (Fig. 37, right).

Adjustable Point: When a radius or the center of an arc is modified, it is usually also necessary to modify the location of one of the arc definition points. An adjustable point enables one to choose a point whose coordinates are to be changed.

73
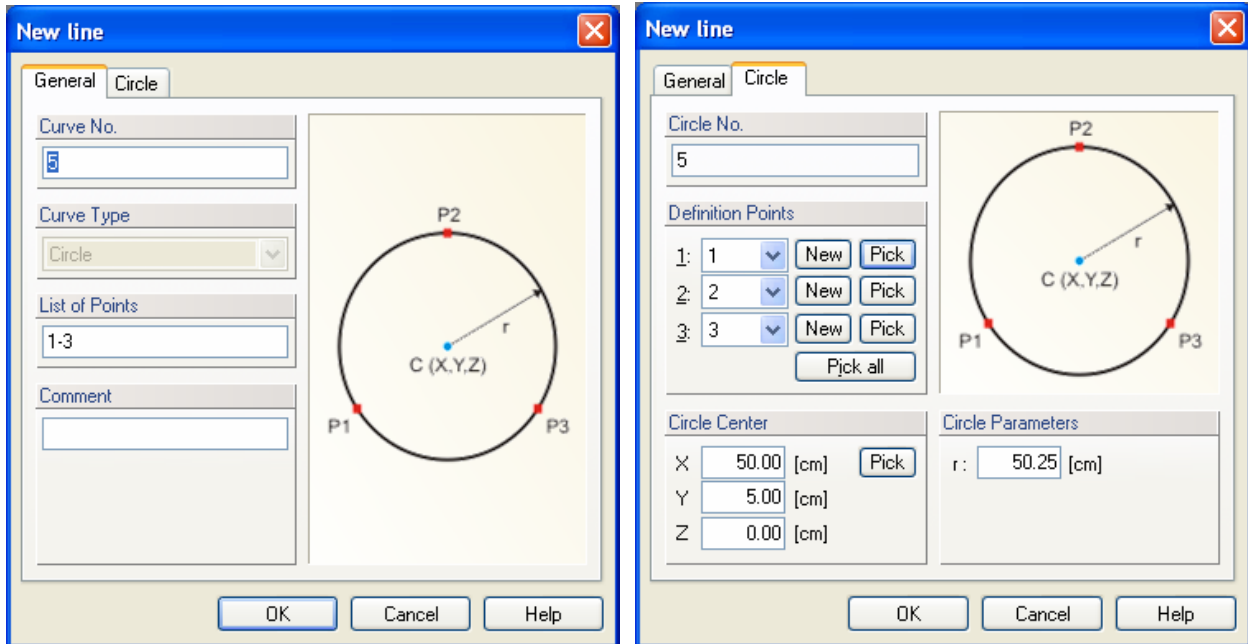
Figure 37. The New Line (Circle) dialog window.

4.1.4. *Curves and Splines*

A spline is a set of more than 2 points connected smoothly by cubic arcs. HYDRUS allows three types of splines:

- cubic spline, i.e., a curve defined by multiple polynomials of the third order. It passes smoothly through all points.
- Bezier's curve – a smooth curve that passes through boundary points, but does not have to pass through internal points .
- B-spline – a general Bezier's curve.

More information can be found at
http://mathworld.wolfram.com/BezierCurve.html
http://mathworld.wolfram.com/B-Spline.html

4.1.5. *Move, Copy, Rotate and Mirror Operations*

All boundary objects can be manipulated using the **Move**, **Copy**, **Rotate** or **Mirror** operations in the **Move – Copy** (Fig. 38), **Rotate** (Fig. 39, left) and **Mirror** (Fig. 39, right) dialog windows. These commands can be access either from the **Tools Menu** or from the **Transform Object** part of the **Domain Geometry** version of the **Tool Bar** at the right side of the **View Window**. Users first select an object to be manipulated, then click on the command, and specify the **Vector of Translation** for the **Move** or **Copy** operations, or the **Angle of Rotation** for a **Rotation**, or define the **Mirroring Plane/Axis** for a **Mirroring** operation.
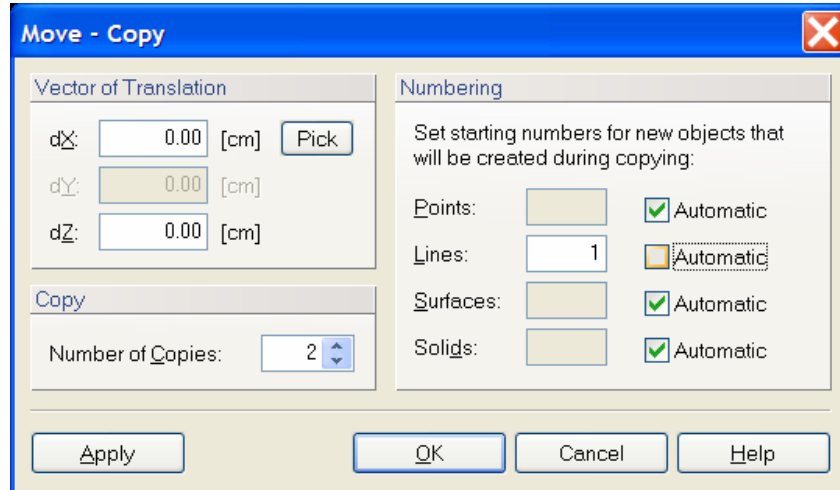
Figure 38. The Move - Copy dialog window.
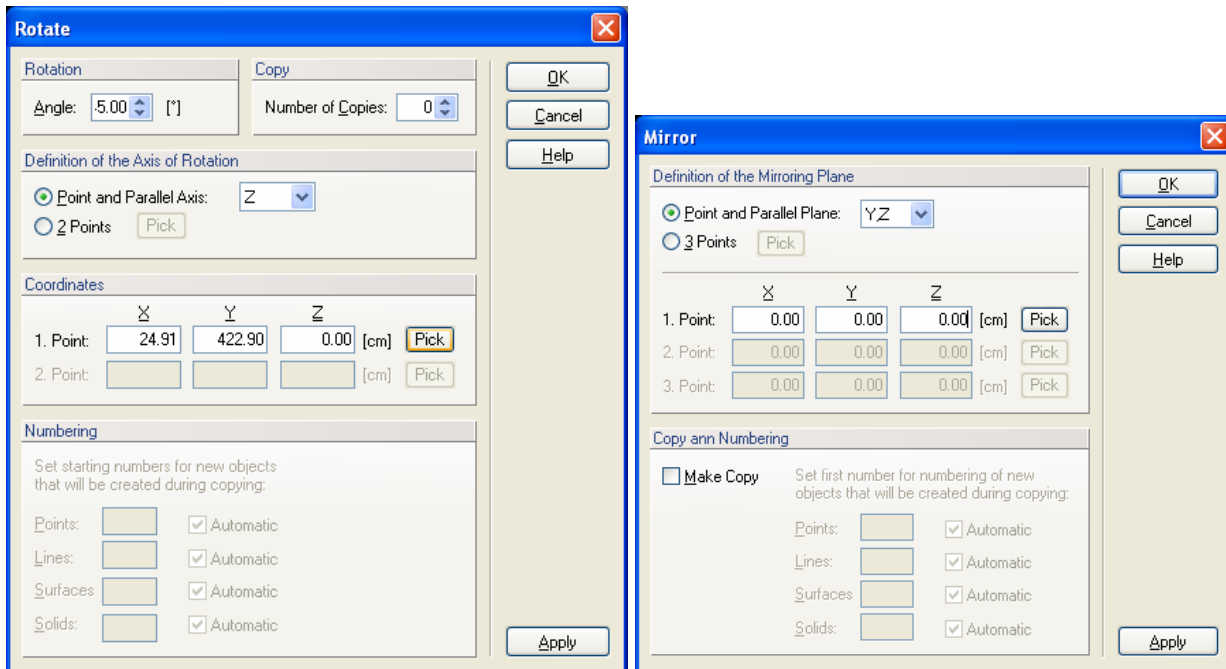
Figure 39. The Rotate (left) and Mirror (right) dialog windows.

75

4.1.6. *Additional Operations*

Additional operations that can be used to manipulate boundary objects are **Intersect Lines**, **Insert Points on Line**, and **Split Line**. All three commands again can be accessed either from the **Tools** menu or from the **Transform Object** part of the **Domain Geometry** version of the **Tool Bar** on the right side of the **View Window**. The first command (**Intersect Lines**) finds the **Intersect** of two lines, whereas the second command **Insert**s **Points on** a **Line**. This can be done either graphically or numerically by specifying the number of nodes to be inserted on the line or distance of the point from the beginning of the line (Fig. 40). The third (**Split Line**) command can be used to split a line into two or more parts.
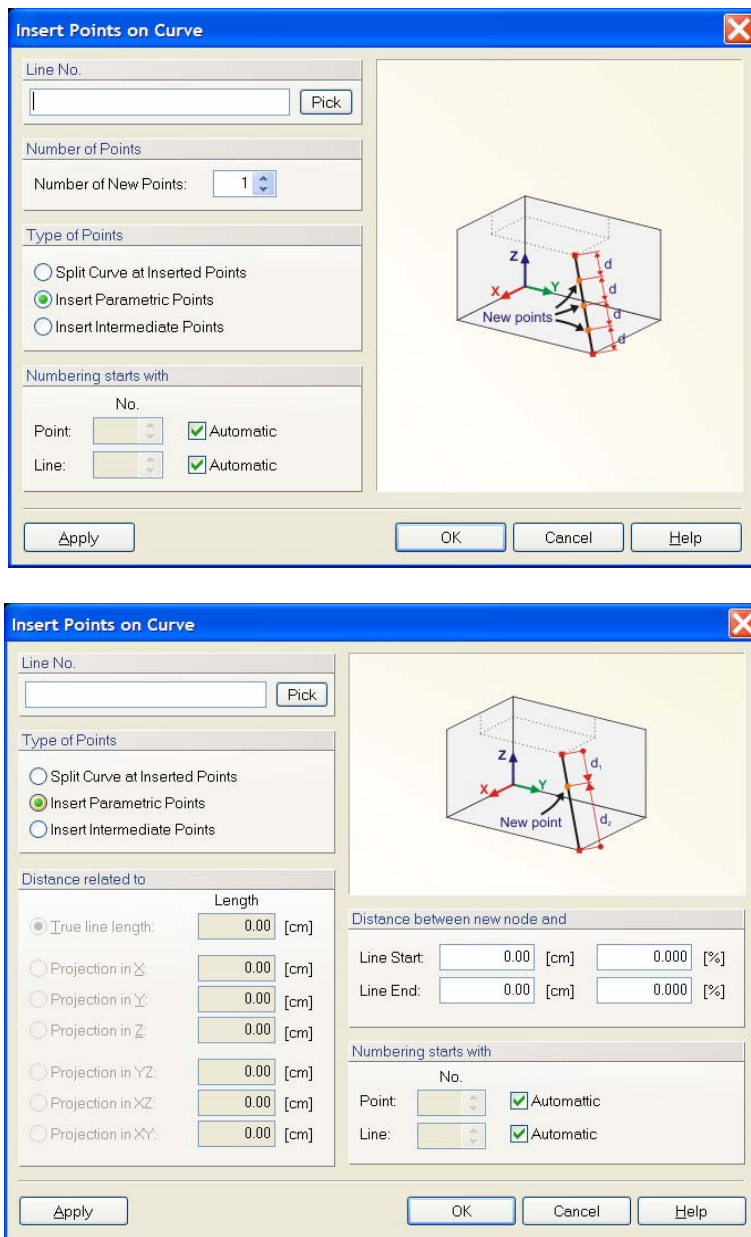
Figure 40. The Insert Point on Curve dialog window.

76

4.2. *Surface*

A **surface** is a closed two-dimensional domain that is either the computational domain for two-dimensional applications or the **base surface** that can be extended into a **solid** for three-dimensional applications. A **surface** is defined by the **List of Boundary Curves**. It can be created using either the *Insert->Domain Geometry->Surfaces->Graphically* or *Insert->Domain Geometry->Surfaces->Graphically Rectangle* commands. Alternative commands on the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar** are **Surface by Rectangle** and **Surface by Boundaries**. In the first case, a cursor appears and users can create a rectangular surface using the mouse. In the second case, users can create a surface by clicking on a closed curve (one or more boundary objects forming a close curve). A **surface** can be edited using the **Edit Surface** dialog window (Fig. 41), which specifies surface type, the number of boundary curves defining the surface, its number, and had a box for possible comments or a description. A **surface** must be created before one can do finite element discretization.
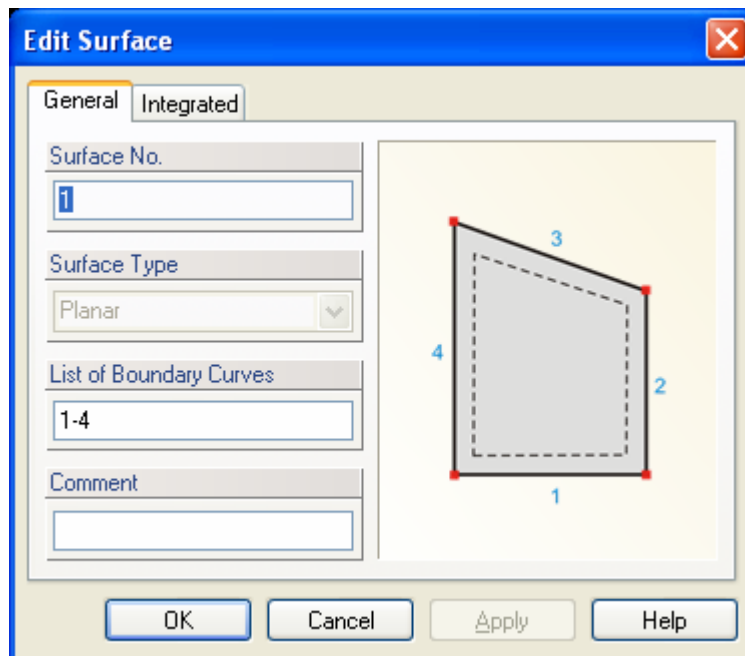
Figure 41. The Edit Surface dialog window.

A **base surface** is formed by a planar surface of arbitrary shape. The base surface can contain **openings**, **internal curves**, and **internal points** (Fig. 42). In the current version of HYDRUS, a base surface can be formed by a single surface. Future versions will permit multiple surfaces to form one single base surface.
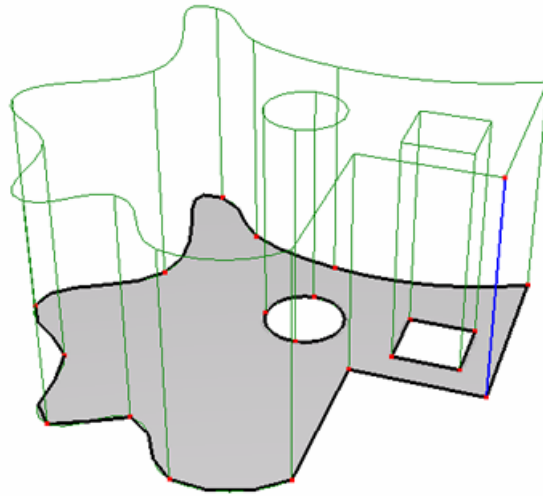
Figure 42. A solid showing the base surface.

If the solid needs to be divided into vertical columns, then these columns must be defined using internal curves in the base surface. The FE-mesh then follows exactly the specified shape of these internal curves (Fig. 43):
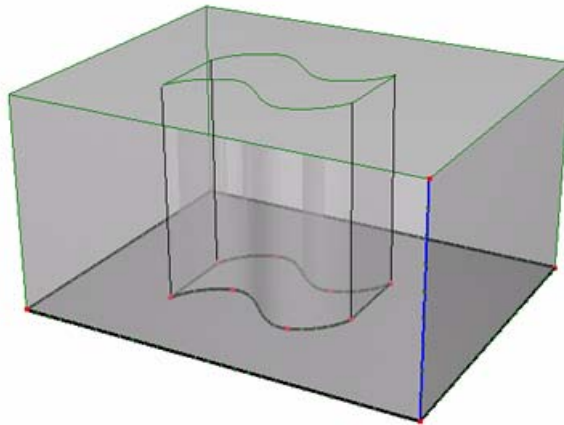
Figure 43. Solid showing separate vertical columns.

A base surface can be defined by a plane other than the horizontal plane, while thickness vectors can be defined in other than the vertical direction. Figures 44 and 45 show an example of a solid that has a base surface in the XZ plane and thickness vectors in the Y direction.
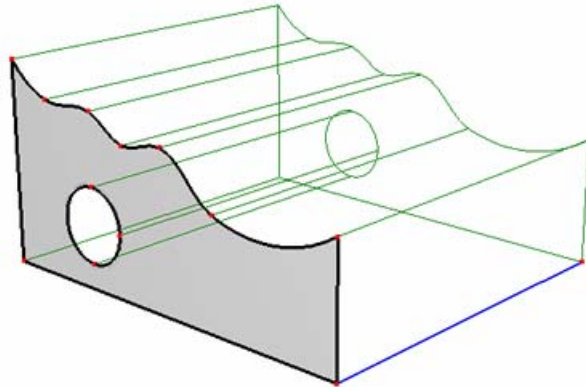
Figure 44. A solid with its base surface in the XZ plane and thickness vectors in the Y direction.
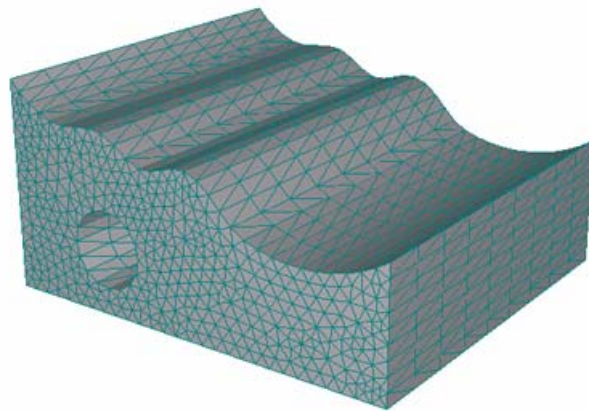
Figure 45. FE-Mesh for a solid with its base surface in the XZ plane and thickness vectors in the Y direction.

4.3. *Opening*

An **opening** is an internal hole defined by a **boundary curve** having the following properties: the curve is closed, positively oriented (in a counter-clockwise direction), does not intersect any other curve or itself, and has the computational domain located on its right-hand side (in the sense of positive orientation), while the left side is not part of a computational domain. An **opening** is not part of the computational domain or surface. An **opening** is created graphically as follows: One first defines boundary objects that create a closed boundary curve. One then uses the command *Insert->Domain Geometry->Opening->Graphically* from the menu, or alternatively the command **Opening by Boundaries** on the **Insert Object** part of the **Domain Geometry** version of the **Tool Bar**. A user creates an **opening** by clicking on the closed curve (one or more boundary objects forming a closed curve). Alternatively, an **opening** can be created using the **New Opening** dialog window (Fig. 46) by clicking on the command *Insert->Domain Geometry->Opening->Dialog* from the menu. An **opening** can be edited using the **Edit Curve** dialog window (Fig. 35), which specifies the number boundary curves defining the opening, its number, and has a box for possible comments or a description.
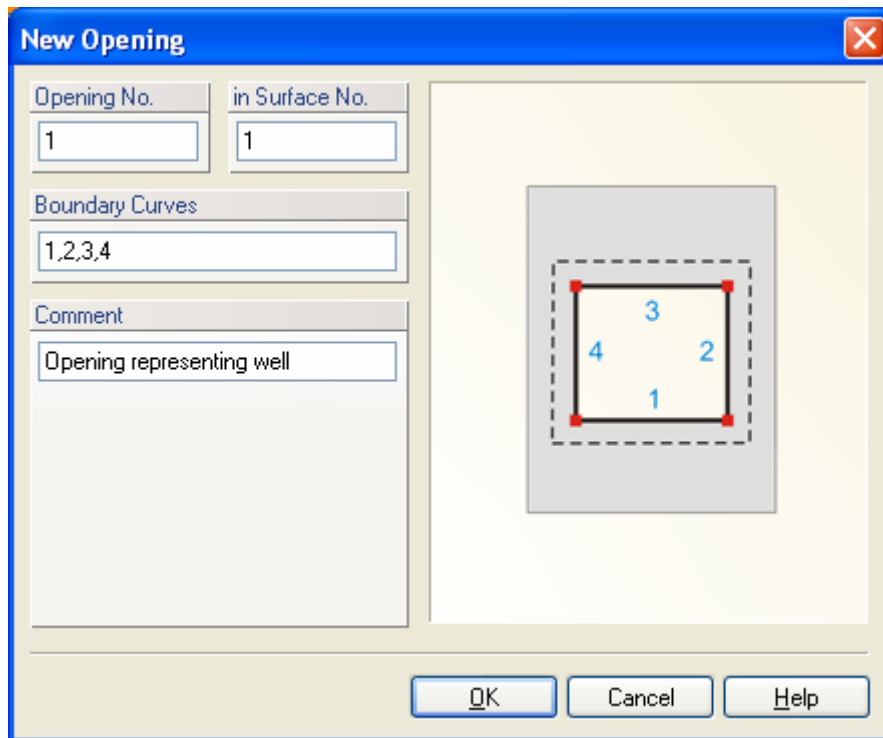
Figure 46. The New Opening dialog window.

4.4. *Solids*

Solids are three-dimensional objects defined by the **base surface** and one or more **thickness vectors**. Thickness vectors do not have to be perpendicular to the **base surface**. A **Solid**, i.e., its **base surface** and **thickness vectors**, is defined and can be edited in the **Edit Solid** dialog window (Fig. 47) that has three tabs: **General**, **Layers**, and **FE-Mesh**. The **General Tab** provides information on which **base surface** and which **thickness vectors** define the solid. The **Layers Tab** informs whether the solid is divided into one or more **Layers**. **Layers** are other objects that can be used to subdivide a single solid. These layers can be used, for example, to keep constant thicknesses of selected horizons or constant discretization close to the soil surface (to get good estimates of evaporation) across the entire transport domain (solid). Layers can be specified using **Profiles**. One profile is created by the code automatically. Users can then define one or more profiles that are associated with different **Thickness Vectors**. These profiles can then be subdivided into multiple layers that can have either constant or variable thickness (**T**) across the transport domain. Thicknesses and the mode (constant or variable) of particular layers are specified in a table. Depth (**D**) is then calculated by adding thicknesses of particular layers. At least one layer thickness must be variable. The finite element discretization then follows these layers. Finally, the **FE-Mesh Tab** (Fig. 48) specifies how many horizontal FE-Layers are used to discretize the solid. When only one layer exists then users can specify relative finite element spacing (spacing of vertical discretization layers) on the vertical side (*FE-Mesh Layer Spacing*) using the RS1 (relative size at the top) and RS2 (relative size at the bottom) factors below *Generate Mesh Layer Spacing*. The element sizes are then proportionally distributed. The preview part of the dialog window shows the main terms used on each Tab. When multiple layers exist then users can specify relative sizes of elements for each layer (*FE-Mesh Density in Layers*).
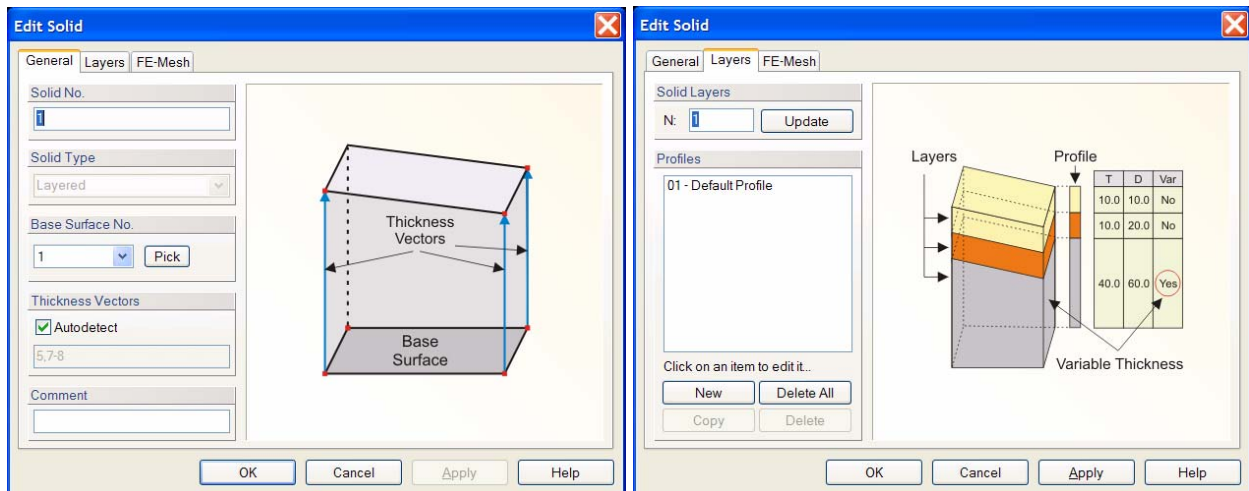
Figure 47. The Edit Solid dialog window; the General and Layers Tabs.
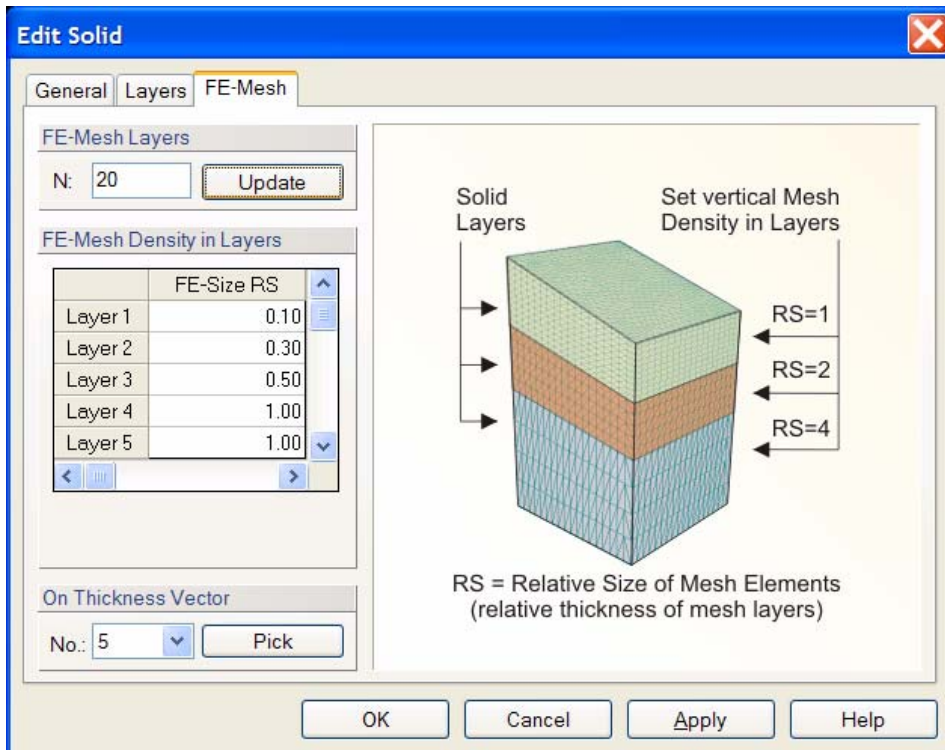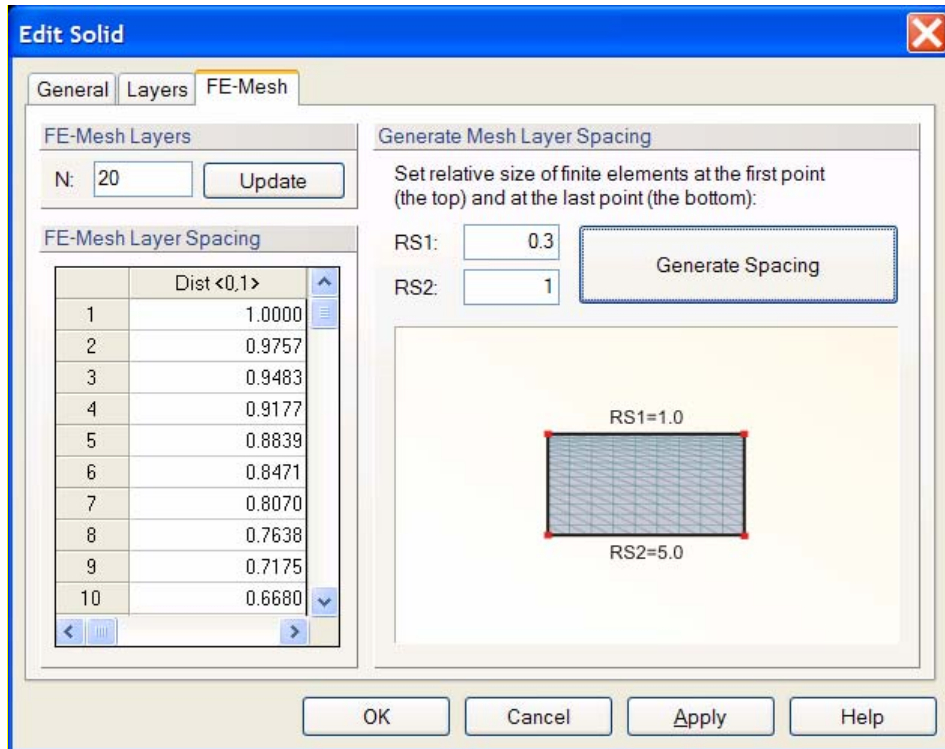
Figure 48. The Edit Solid dialog window; the FE-Mesh Tab for a single and multiple layers.

4.5. *Thicknesses*

The term **Thickness Vector** is used for a vector (usually but not always perpendicular to the **Base Surface**) that extends the Base Surface to form a **solid** (the three-dimensional **computational domain**). A definition of the **Thickness Vector** (i.e., its **Boundary Points**), is given and can be edited in the **Thickness** dialog window (Fig. 49).
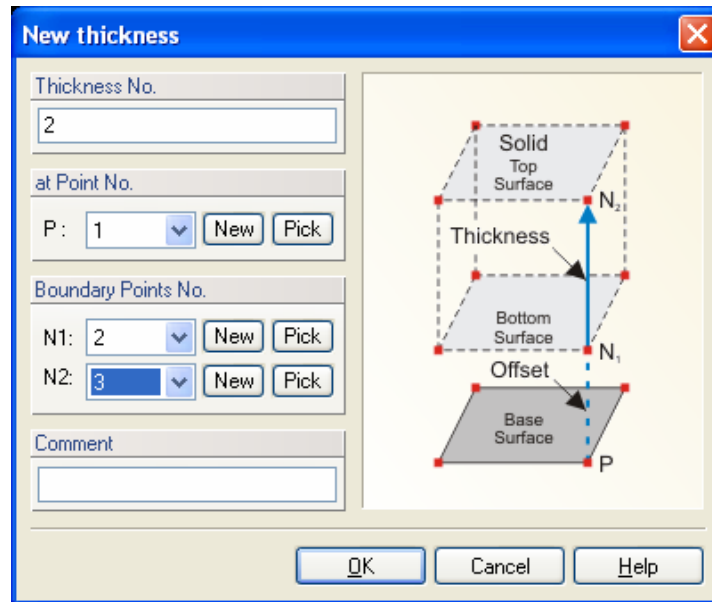
Figure 49. The Thickness dialog window.

The height of a solid is defined using one or more **Thickness vectors**. Each thickness vector is defined by an **Anchor Point P** and two **Boundary Points N1** and **N2**. The anchor point P must be part of the base surface, i.e., it must be either a defining point of the external boundary or the internal curve, or an internal point in the base surface. Boundary points N1 and N2 are arbitrary points in 3D space. Coordinates of these points can be edited, thus allowing one to specify the thickness vector in an arbitrary direction (i.e., not necessarily perpendicular to the base surface). Usually the anchor point P is the same as boundary point N1 so that one can use the same index for both P and N1. If, for whatever reason, we do not want to have the base surface on the bottom of the transport domain (Solid) (e.g., when the bottom of the transport domain is not in the same plane), users can make the N1 node be different from P (the red part in Fig. 50).
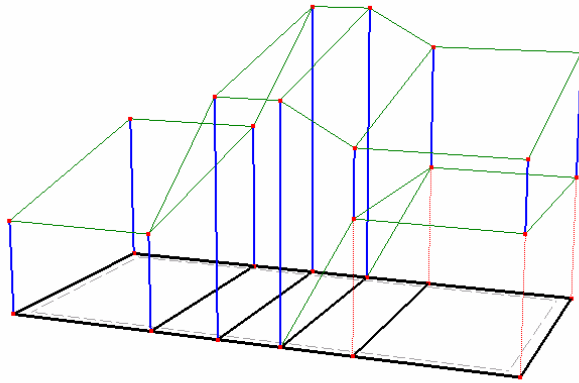
Figure 50. A solid with several thickness vectors.
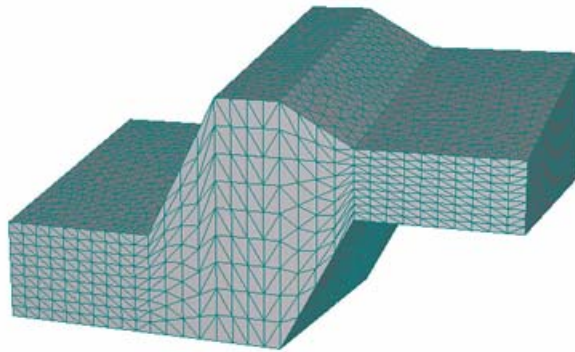
Figure 51. FE-Mesh for the solid in Figure 50.

The height of a solid is constant when less than three thickness vectors are used. Three thickness vectors define a linear plane (with generally an inclined surface). More than three thickness vectors with different lengths then define the top surface that is formed by triangles, whose coordinates are calculated from the thickness vectors using linear interpolation or extrapolation.

When breaks in the slope of the top surface are to be defined exactly, then it is necessary to define internal curves in the base surface. Figures 52 and 53 show the importance of having internal curves (or not having them) in the Y direction for proper definition of the solid (compare Figures 50 and 52 Figures 51 and 53).
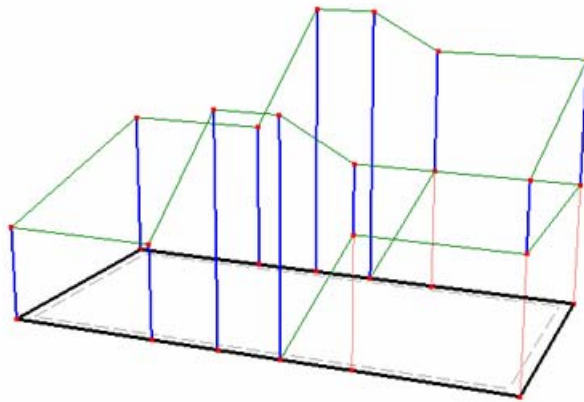
Figure 52. Missing internal curves in the base surface.
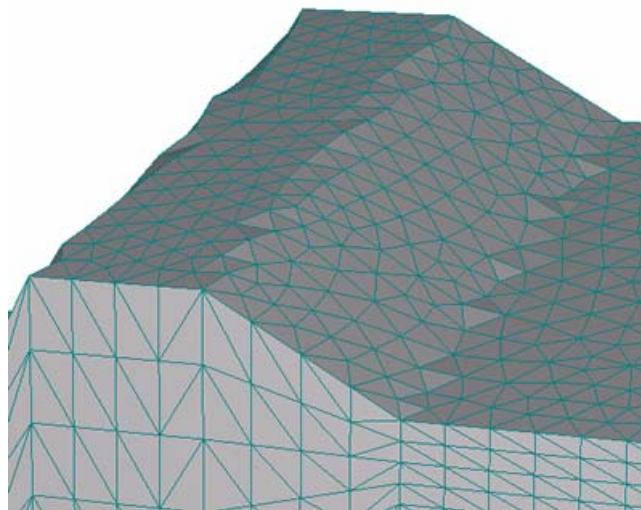
Figure 53. Consequence of missing an internal curve in the base surface on the FE-Mesh of the top surface.

4.6. *Auxiliary Objects*

In addition to objects that define the computational domain, the HYDRUS GUI allows users to employ several **Auxiliary Objects** that can be used to, for example, add **Dimensions** to the computational domain, include various **Labels**, or define permanent **Cross Sections** or **Mesh-Lines**.

4.6.1. *Dimensions*

Dimensions can be added to describe spatial properties of the computational domain in the **View** window using the *Insert->Auxiliary Objects->Dimensions* command or the **Dimensions** command from the **Insert Object** part of the **Domain Geometry** version of the **Edit Bar**. Then one needs to click at two points defining the computational domain and drag **Dimensions** to the required position. Figure 64 shows an example of how **Dimensions** can be used.

4.6.2. *Labels*

Labels can add any desired text to the computational domain in the **View** window using the *Insert->Auxiliary Objects->Dimensions* command or the **Comments** command from the **Insert Object** part of the **Domain Geometry** version of the **Edit Bar**. One then clicks simply anywhere in the **View** window and write the desired text. The text itself, its color, frame, and its offset can be specified in the **Edit Comment** dialog window (Fig. 54). Figure 64 shows an example of how the "Furrow" **Label** is used.
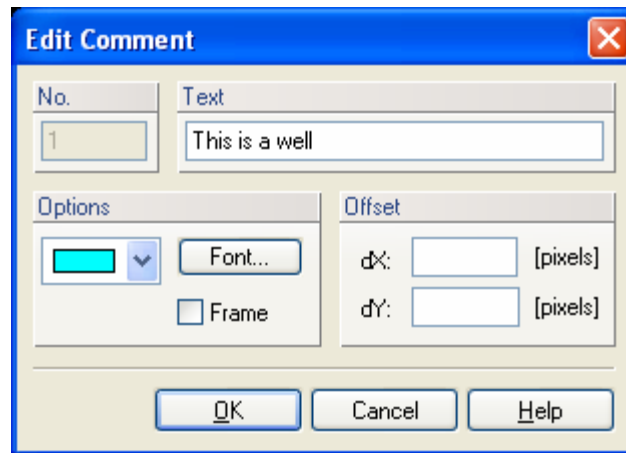
Figure 54. The Edit Comment dialog window.

4.6.3. *Bitmaps* (*Textures*)

Bitmaps (**Textures**) serve to use scanned figures (maps) as means to define the computational domain in the **View** window. **Bitmaps** can be added using the *Insert->Auxiliary Objects->Textures* command. Corners of the **Bitmap** must be anchored at 4 points, coordinates of which

must be selected such that the scale of the Bitmap corresponds with the scale of the View window. One can then simply trace the bitmap to specify the computational domain.

4.6.4. *Cross-Sections*

In the HYDRUS-2D software package one could click at any two points of the transport domain to display results of selected variables between those two points, i.e., along a specified cross-section. The exact location of these cross-sections was not saved and they had to be redefined whenever a new graph was required. In HYDRUS one can define the cross-sections and save their locations so that graphs along the cross-sections can be recalled at any time by simply clicking at them. Graphs along pre-defined cross-sections can be display for both the initial conditions and the output results. For example, if a plot of the pressure head along a predefined cross-section at a particular time is needed, one needs to display the pressure head outputs, find a particular time, and then click on the predefined cross-section. The graph is displayed instantaneously. Specifying the cross-section within a two-dimensional domain is straightforward. For three-dimensional domains one can use the **Cross-Section** dialog window (Fig. 55) for this purpose.
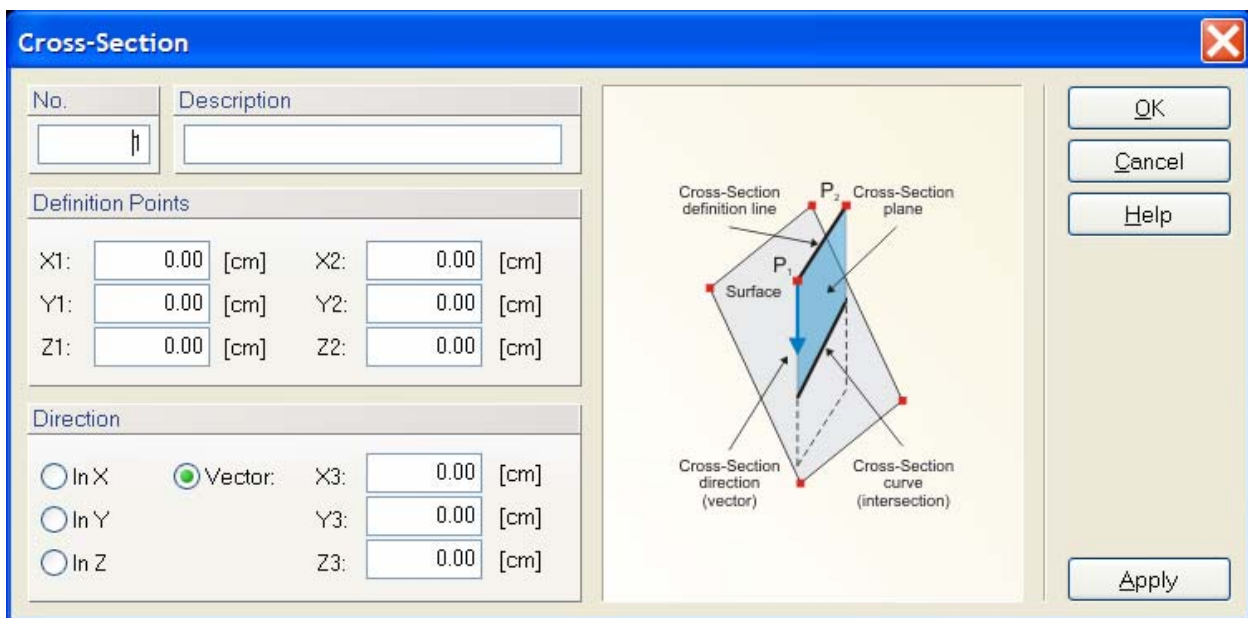
Figure 55. The Cross-Section dialog window.

4.6.5. *Mesh-Lines*

Mesh-Lines are very similar to **Cross-Sections**, except that **Mesh-Lines** follow edges of the finite elements and do not have to be straight. They are used, similarly to **Cross-Sections**, to display selected variables along defined **Mesh-Lines.** Similarly to **Cross-Sections**, the locations of **Mesh-Lines** are saved and can be recalled at any time.

5. Finite Element Mesh

5.1. *Finite Element Mesh Generator*

The **Finite Element Mesh Generator** dialog window (Fig. 56) is used whether to select a structured finite element mesh for relatively simple rectangular or hexahedral domain, or a more general unstructured finite element mesh. The dialog provides a brief description of each mesh generator and a simple bitmap with an explanation of the main terms involved. While the structured finite element generator can be used only for simple rectangular of hexahedral domains, the unstructured finite element generator is used for more complicated geometries.
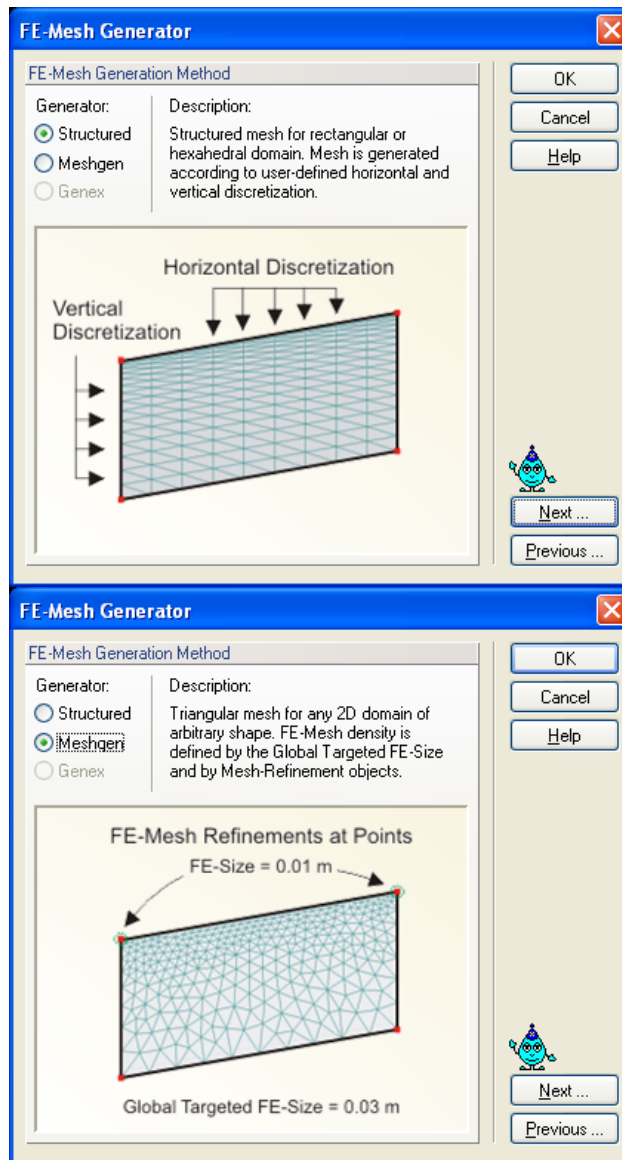
Figure 56. The Finite Element Mesh Generator dialog window. One version of the dialog for the structured FE mesh is shown at the top and one for the unstructured mesh at the bottom of the window.

5.2. *Structured Finite Element Mesh Generator*

As discussed in Section 2, two-dimensional transport domains can be defined using modified rectangles. Simple rectangular domains are defined by three straight lines, one at the bottom of the domain and two at the sides, while the upper boundary may or may not be straight (see examples in Fig. 8). Nodes along the upper boundary may in that case have variable *x*- and *z*-coordinates, but the lower boundary line must be straight (horizontal or with a specified slope), whereas the left and right boundary lines must be vertical. The flow region can then be discretized into either a structured or an unstructured triangular finite element mesh.

When the structured mesh is used one then needs to specify in the **Rectangular Domain Discretization** dialog window the number of nodes (*Count*) on the horizontal (*Horizontal Discretization in X-Direction*) and vertical (*Vertical Discretization in Z-Direction*) sides of the rectangular region, including their nodal coordinates (Fig. 57). The *relative size of finite elements* on the vertical side can be modified using the RS1 (relative size at the top) and RS2 (relative size at the bottom) factors below *General Vertical Coordinates*. The element sizes are then proportionally distributed.

The upper boundary is by default parallel with the bottom boundary. Any possible vertical deviations from this parallel line can be defined using *dz* values (in the *Horizontal Discretization in X-Direction* part of the window). Relatively general domains can still be defined by properly adjusting the *dz* values (see Fig. 8).
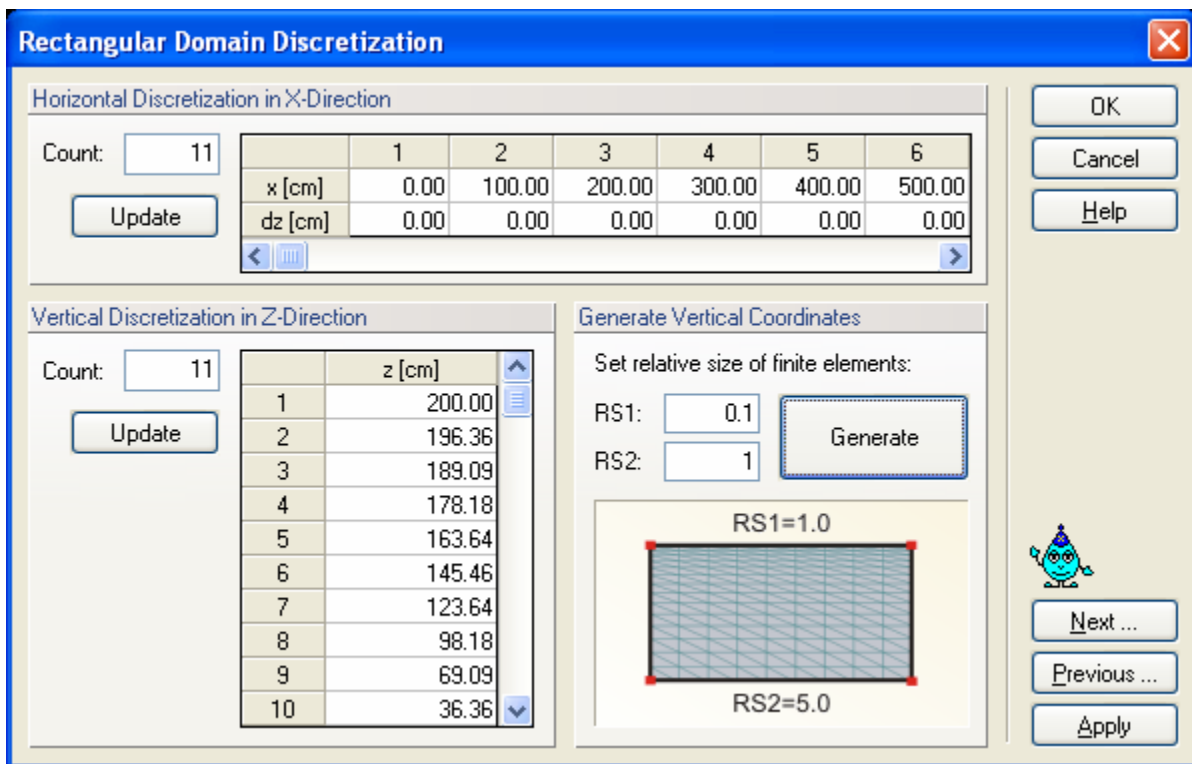
Figure 57. The Rectangular Domain Discretization dialog window.

89

A similar approach can be used to discretize simple hexahedral domains in three dimensions. Hexahedral domains must have similar properties as rectangular domains in that they are defined by vertical planes at the sides, a horizontal plane (possibly with a certain slope) at the bottom boundary, and with only the upper boundary not having to be a plane. The discretization of the hexahedral domain is then defined in the **Hexahedral Domain Discretization** dialog window. Again, one needs to specify the number of nodes (*Count*) on the horizontal (*Horizontal Discretization in X, Horizontal Discretization in Y*) and vertical (*Vertical Discretization in Z-Direction*) sides of the hexahedral region and their nodal coordinates (Fig. 58). The *relative size of finite elements* on the vertical side can again be modified using RS1 (relative size at the top) and RS2 (relative size at the bottom) factors (*General Vertical Coordinates*). The vertical final element sizes are then proportionally distributed.

Any possible vertical deviations from the plane parallel with the bottom of the domain can be defined using the *dz* values (in the *Horizontal Discretization in X* and *Horizontal Discretization in Y* parts of the window). This feature still allows relatively general domains too be created (see Fig. 9).
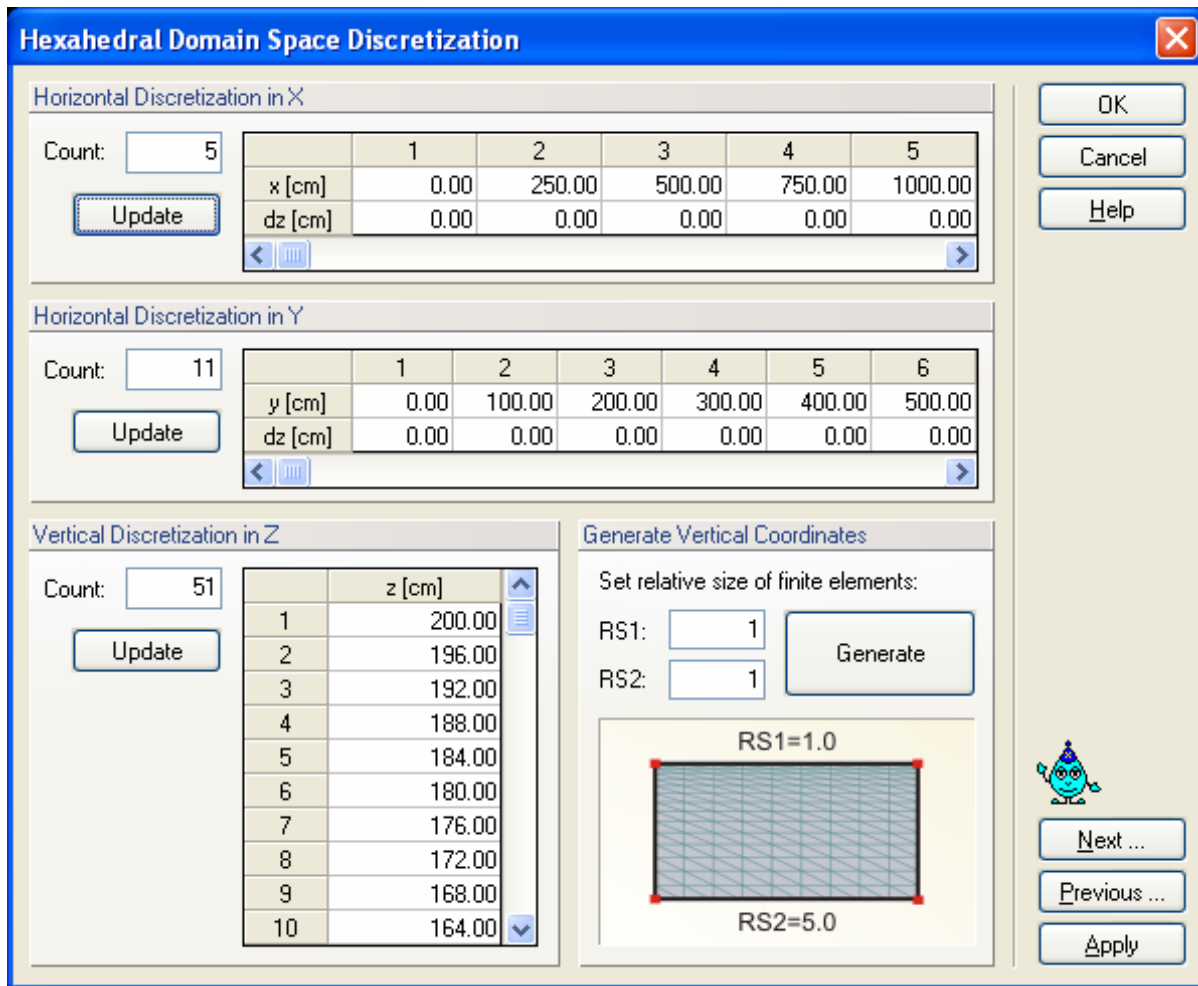
Figure 58. The Hexahedral Domain Discretization dialog window.

90

5.3. *Unstructured Finite Element Mesh Parameters*

Parameters for generating the *Unstructured Finite Element Mesh* are specified in the **FE-Mesh Parameters** dialog window (Figs. 59 through 62). This dialog window has four Tabs in which various parameters of the unstructured finite element mesh can be specified.

The Main Tab

The *Targeted FE size* (i.e., the average size of the triangular elements in the generated finite element mesh) is specified on the **Main** Tab (Figs. 59). The program selects by default a **Targeted FE Size**. Users can change this value by deselecting the **Automatic** check box. The finite element mesh with this *Targeted FE size* can be further modified using various tools, such as *Stretching* in different directions (on the **Stretching** Tab, Fig. 60) to make the mesh anisotropic, specifying the *Maximum Number of Nodes on Boundary Curve* (on the **Options 1** Tab, Fig. 61) and *Minimum Number of Nodes on Boundary Curve* (on the **Options 2** Tab, Fig. 62), and using **Finite Element Mesh Refinement** (Fig. 63). While the **Page Default** command sets default values on a particular tab of the **FE-Mesh Parameters**, the **All Default** command sets default values on all four tabs. A user can specify on the **Main** Tab the **No. of Horizontal Layers** (only for three-dimensional applications), which are layers parallel with the **Base Surface** to add the third dimension to the problem.
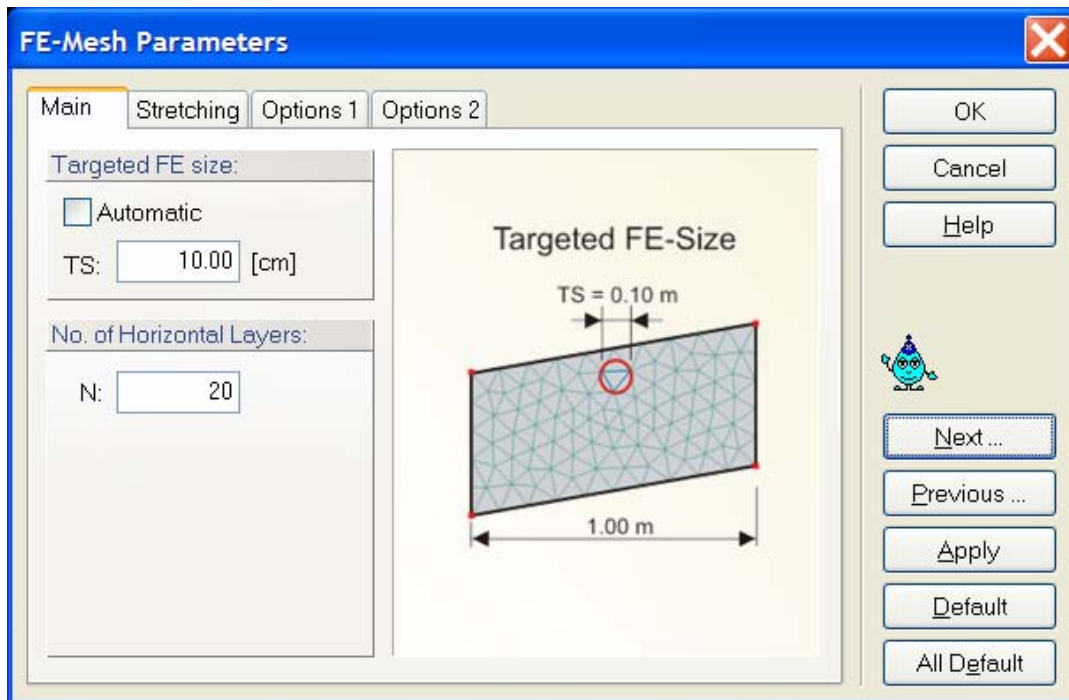
Figure 59. The FE-Mesh Parameters dialog window (Tab Main).

The Stretching Tab

Stretching of the finite element mesh (i.e., the degree of mesh anisotropy in a certain direction) is defined using the **Stretching Factor** and **Stretching Direction** (Fig. 60). The finite elements are made larger in the particular **Stretching Direction** if the **Stretching Number** is larger than one, and smaller if smaller that one. The result of this transformation is a mesh deformed in the given direction, which can be desirable for problems that, for example, require different spatial steps (mesh sizes) in the X and Y directions. The **Stretching Direction** is defined by a vector with two coordinates (Vx and Vz) for vertical two-dimensional domains. For three-dimensional problems this vector requires three components (Vx, Vy, and Vz).
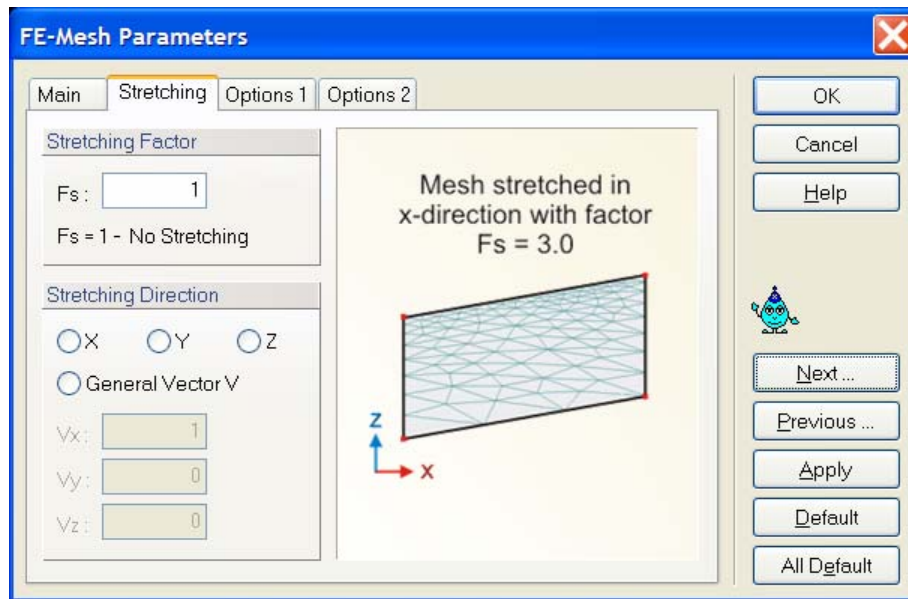
Figure 60. The FE-Mesh Parameters dialog window (Tab Stretching).

The Option 1 Tab

Parameters for the unstructured triangular finite element generator are given in the **Options 1** Tab of the **FE-Mesh Parameters** dialog window (Fig. 61). The parameters are divided into *FE-Mesh Limits* (which limits the number of elements) and *FE-Mesh Quality* (which affects the smoothness of the FE mesh) groups.

The following parameters are specified in the *FE-Mesh Limits* group:

Maximum Number of Nodes on Boundary Curves: This is the maximum total number of nodes on all boundary curves for two-dimensional applications, or on all boundary curves defining the bottom plane (base surface) for three-dimensional applications.

Maximum Number of FE-Mesh Nodes (2D Mesh): This is the maximum total number of finite element nodes in two-dimensional domains, or on the bottom plane (base surface) of three-dimensional domains.

Both parameters are mainly informative and may lead to an interruption of the FE-mesh generation process. The mesh generation is interrupted by the message**: *Achieved the maximum number of nodes!* This means that the maximum allowed number of nodes (either on the boundary curves or in the two-dimensional domain) was reached during the mesh generation process. This is usually a consequence of having too many nodes along the boundaries (the number of mesh nodes inside a domain increases approximately with the square of the number of boundary nodes). It is then necessary to decide whether or not so many nodes are needed for the envisioned triangular mesh. If the answer is 'yes' then the maximum number of nodes must be increased in this dialog window. If the answer is 'no' then it is necessary to decrease the **Targeted FE Size** (Figs. 59) or to increase the **Smoothing Factor** (in the *FE-Mesh Quality* group discussed below).
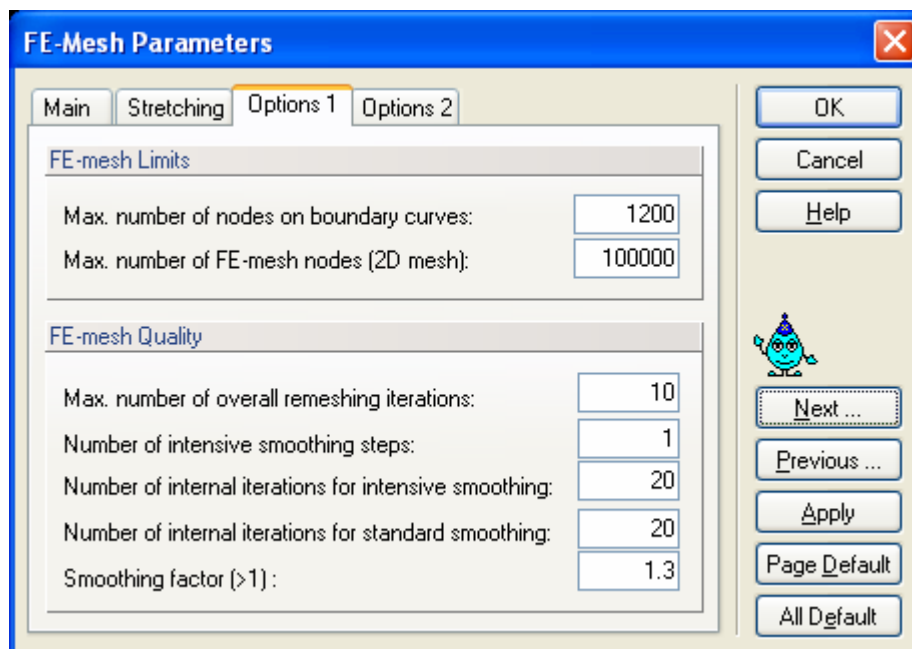
Figure 61. The FE-Mesh Parameters dialog window (Tab Options 1).

The following parameters are specified in the *FE-Mesh Quality* group:

Maximum Number of Overall Remeshing Iterations:
This number defines the maximum number of iterations during finite element mesh remeshing. In most cases the resulting mesh is obtained within fewer iterations than the default value of 10. In some cases the repeated adding and removing of nodes can cause an infinite loop. In that case (or when the mesh generation process converges very slowly) the code terminates after reaching the maximum number of iterations as defined by this value.

Number of Intensive Smoothing Steps:
Intensive smoothing repeats the operations of Delaunay remeshing and smoothing until there are no more changes during the Delaunay remeshing step. This parameter specifies the number of intensive smoothing cycles in the beginning of the mesh generation process, which can significantly influence the mesh smoothness. However, too many smoothing cycles can significantly slow down the mesh generation process. The recommended value is between 1 (less smoothing) and 3 (more smoothing).

Number of Internal Iterations for Intensive Smoothing:
This number defines the maximum number of iterations during one intensive smoothing step. This number guaranties that intensive smoothing will stop after a specified number of iterations, even when the prescribed criterion is not reached (i.e., some changes would still occuer during Delaunay remeshing).

Number of Internal Iterations for Standard Smoothing:
This number defines the maximum number of iterations while solving the elliptic equations, a process needed during mesh smoothing; it significantly influences the final smoothness of the mesh. A higher number of iterations improves the mesh smoothness. It serves little purpose to increase the number above 20 since the mesh is then virtually constant anyway, while the whole process of mesh generation would be slowed down significantly.

Smoothing Factor:
The smoothing factor is the ratio of the maximum and minimum height of a finite element triangle. For a triangle with equal sizes this factor is equal to 1 (which is theoretically not achievable for finite element meshes). The smoothing factor can be decreased to a value of about 1.1 when a highly smooth finite element mesh is required and, vice-versa, can be increased when a course mesh can be tolerated. The smoothing factor significantly affects the final number of elements.

In general, the default values in the FE-Mesh Parameters dialog window should be preserved; only experienced users should modify the various parameters needed for the mesh generation process.

The Option 2 Tab

Minimum Number of Points on Each Closed Boundary Curve is set by default equal to 15. This number can be changed in the **FE-Mesh Parameters** dialog window of the **Options 2** Tab (Fig. 63). This parameter is important when such objects as openings representing wells or drains are included in the transport domain. These objects may be very small compared to the global finite element mesh (i.e., smaller than the targeted finite element size). Having a minimum number of nodes on boundary curves will then lead to local refinement of the finite element mesh around these objects, thereby ensuring that relatively small objects are accurately represented in the numerical solution.
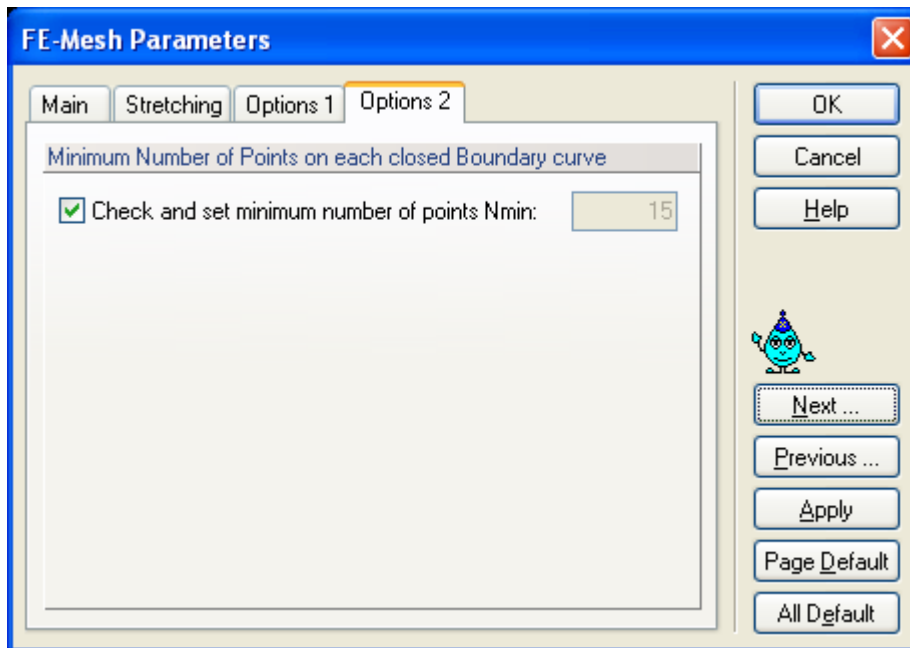
Figure 62. The FE-Mesh Parameters dialog window (Tab Options 2).

5.4. *Finite Element Mesh Refinement.*

The Finite Element Mesh Refinement is carried out in two steps:

1. One must first define the desired type of **FE Mesh Refinement** using the dialog window shown in Figure 63. Mesh refinement can be defined around **Points**, **Lines**, **Number of Points on the Line**, or **Surfaces**.

2. Next, users must assign the refinement to particular Points, Lines or Surfaces, so the program knows where the refinement should take place. After double clicking on a particular point, line, arc, circle or spline, the corresponding dialog window will appear (the **Edit Point** dialog window, Fig. 34; or the **Edit Curve** dialog window, Fig. 35), where users should select **FE-Mesh Refinement** in the **FE-Mesh Tab**. The code will then create a list of nodes (or lines or surfaces) for a particular refinement that can be further edited by a user. **FE Mesh Refinement** is graphically displayed using red dots in green circles for nodes, green nodes for lines, and a small square in the corner of a surface. By clicking on these colored items, the **FE Refinement** can be deleted or edited. Editing of the FE mesh refinement will affect all objects to which a particular refinement was assigned.
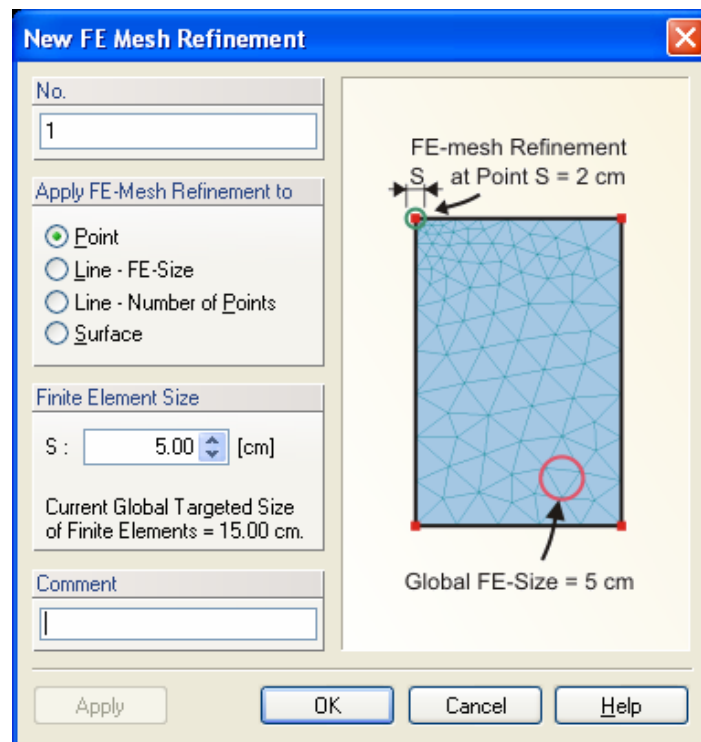
Figure 63. The New FE-Mesh Refinement dialog window.

As an example, Figure 64 shows the transport domain (700 * 650 cm) and the finite element mesh of a problem with a furrow and a drain. The mesh was generated with a **Targeted FE Size** of 50 cm and three **FE-Mesh Refinements** (Fig. 64, top). Refinement 1 (with a finite element size of 10 cm) was assigned to nodes 1, 2, 4 and 5, refinement 2 (5 cm) to node 3, and refinement 3 (15 cm) to nodes 8 and 9. There were 15 nodes on the drain boundary (due to the command **Minimum Number of Points on Each Closed Boundary Curve**). The resulting finite element mesh is shown in Figure 64 (bottom).
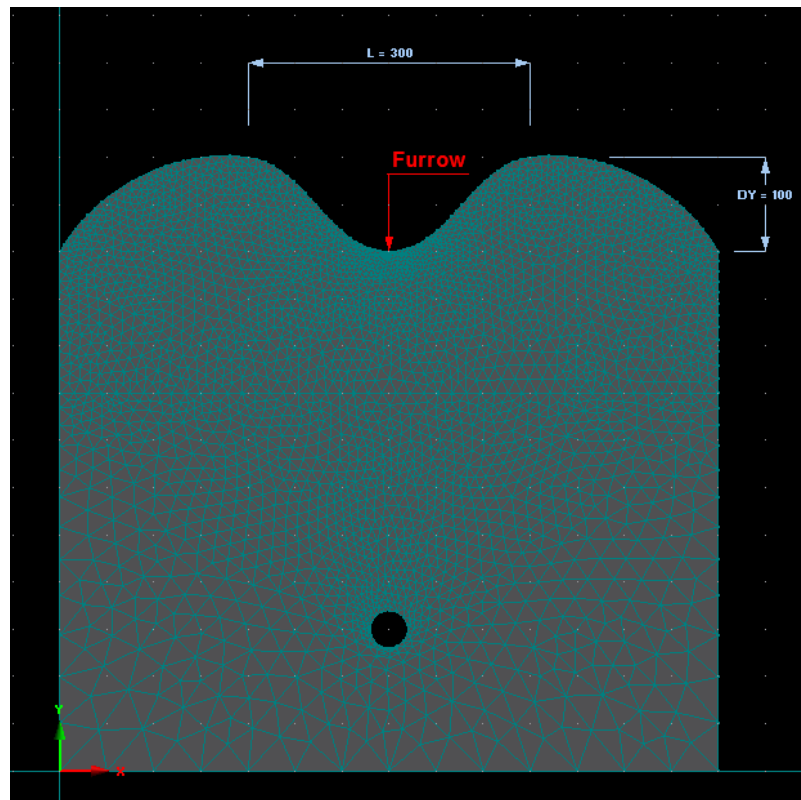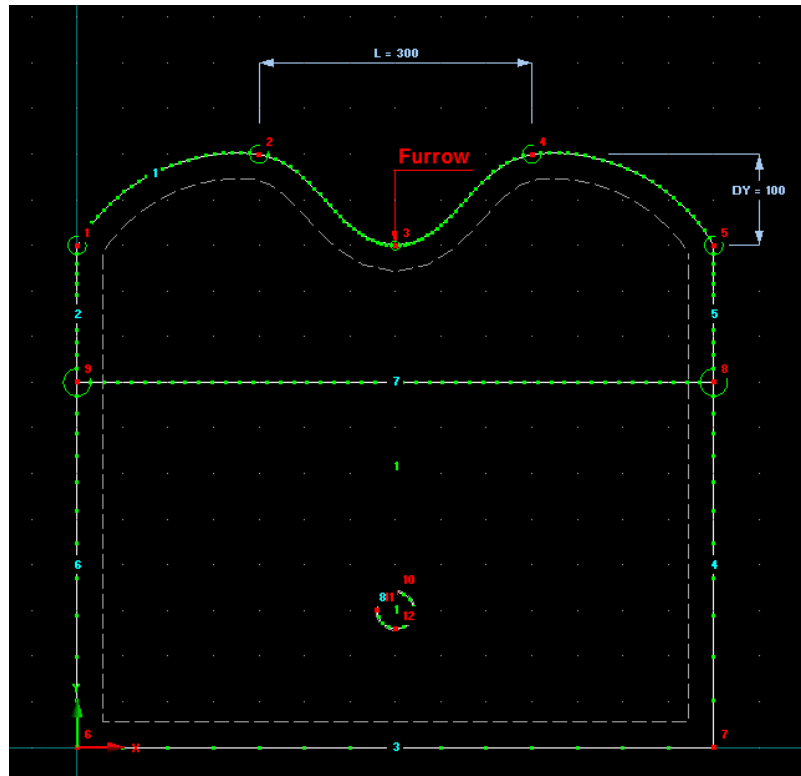
Figure 64. Example of FE-Mesh Refinements (top) and FE-Mesh (bottom).

5.5. *Unstructured Finite Element Mesh Generator MeshGen2D*

The **MeshGen2D** module may be used to discretize a two-dimensional flow region (or a base plane of the three-dimensional domain) into an unstructured triangular mesh. The algorithm used for this purpose is general and can be applied to virtually any two-dimensional computational domain. The first step of the mesh generation process discretizes the boundary curves, while the second step generates the unstructured triangular mesh.

Generation of Boundary Points
The first step of the mesh generation process involves discretization of the boundary curves. During this step boundary nodes are generated on all boundaries and internal curves by dividing them in abscissas (i.e., short boundary edges). If no previous boundary nodes existed, the program automatically generates a default equidistant point distribution. Boundary nodes can be edited by users to optimize the lengths of the boundary edges using the **FE-Mesh Parameters** dialog window (Figs. 59 through 62). The local density of the mesh can thereby be determined in any part of the domain (also taking into account the use of internal curves). There are two ways to obtain appropriate distributions of the boundary nodes, i.e., by (1) specifying the *Targeted FE size* (Figs. 59, the Main Tab), and (2) refining the FE-Mesh (the **FE Mesh refinement** dialog window, Fig. 63).

1) A global **Targeted FE Size** (Fig. 59) is the main variable of the FE-Mesh process. Although a default Targeted FE-Size is specified by the program, this value should be adjusted by users in most applications. The default value is used to generate at least a reasonable initial mesh, even for inexperienced users.
2) The finite element mesh can be adjusted locally in the domain by using **FE-Mesh Stretching** (Fig. 51) or **FE-Mesh Refinement** (Figs. 34, 35, and 63). **FE-Mesh Refinement** can be implemented for various geometrical objects, including **Points**, **Lines** or **Surfaces** (Fig. 63). When several **FE-Mesh Refinements** overlap in one location (such as when the **FE-Mesh Refinement** is defined for both a point and a line), then the FE-Mesh Refinement specified for the lower level objects (i.e., a point rather than a line) is used.

The boundary nodal distribution determines in a very substantial manner the ultimate quality and size of the unstructured finite element mesh. Optimally distributing nodes along the boundaries of relatively complicated domains (e.g., a very irregular anisotropic domain) can be a very difficult problem, and may require some experience.

Table 10. Definition of terms related to the boundary discretization.

Boundary Nodes	Boundary nodes are points, marked by green squares, which discretize boundary curves. These nodes are generated along every boundary curve and are ordered in a counter-clockwise direction (on closed curves). Boundary nodes determine the local densities of the triangular mesh that is

being generated for a given boundary nodal distribution, and are part of the triangular mesh.

Boundary Edges	Boundary edges are abscissas discretizing boundary curves. They connect generated boundary nodes, are oriented in a counter-clockwise direction, and are located on the edge of the mesh.
Fixed Points	Fixed points are points on boundary curves marked by red squares. These points may be used to adjust the local density of boundary nodes (using **FE Mesh refinement**). By default, fixed points are placed on all nodes of polylines and on all object boundary points describing the boundary, but they can be also inserted or deleted at any other point on boundary curves.

Generation of the Unstructured Triangular Mesh

The unstructured triangular mesh is generated by means of five operations:
(1)   discretization of the flow domain into triangles with vertices at given boundary nodes (**Fundamental Triangulation**),
(2)   inserting new points in all triangles which do not fulfill a certain smoothness criterion (**Mesh Refinement**),
(3)   implementation of Delaunay's retriangulation for the purpose of eliminating all nodes surrounded by more than six triangles, as well as to avoid extreme angles (**Remeshing**),
(4)   smoothing of the mesh by solving a set of coupled elliptic equations in a recursive manner (**Smoothing**), and
(5)   correction of possible errors which may appear during smoothing of the finite element mesh (**Convexity Check**).
Operations 2 through 5 are repeated until a prescribed smoothness of the mesh has been achieved.

An unstructured triangular mesh for a given boundary nodal distribution can be generated in two different ways: a step-by-step approach (*Calculation->Advanced FE-Mesh Generation->Fundamental Triangulation* and subsequent commands, or by clicking on the **Fundamental Triangulation** command of the **FE-Mesh Advanced** part of the **FE-Mesh** version of the **Tool Sidebar**), or by using automatic mesh generation (*Calculation->Generate FE-Mesh* or the **Generate FE-Mesh** command on the **Edit FE-Mesh** part of the **FE-Mesh** version of the **Tool Sidebar**). The step-by-step approach should be used only for special cases, and then only by experienced users. Automatic generation (recommended) is a much faster and easier approach. The mesh generation parameters must be specified before the mesh generation process is started. By modifying the mesh generation parameters, users can influence the smoothness of the mesh (**Smoothing Factor**, Fig. 61), its anisotropy (Figs. 60 and 65), computational time, and the possible display of intermediate results, among other features. The most important mesh generation parameter is the "smoothing factor", which can significantly affect the final number of elements. The smoothing factor is defined as the ratio of the minimum and maximum dimensions of a triangle. When a very smooth finite element mesh is required, the smoothing factor should be decreased to about 1.1; when a coarser mesh is possible, the smoothing factor can be increased.

The code saves mesh data, by default, in a binary format (the file MESHTRIA.000). However, users have the option to also save mesh data into a file (MESHTRIA.TXT) in ASCII format (the Program Option dialog window, Fig. 92), and then use possibly the data later for other calculations.

When users do not have authorization to use the MeshGen2D module, they can employ the internal mesh generator for relatively simple rectangular or hexahedral transport domains.
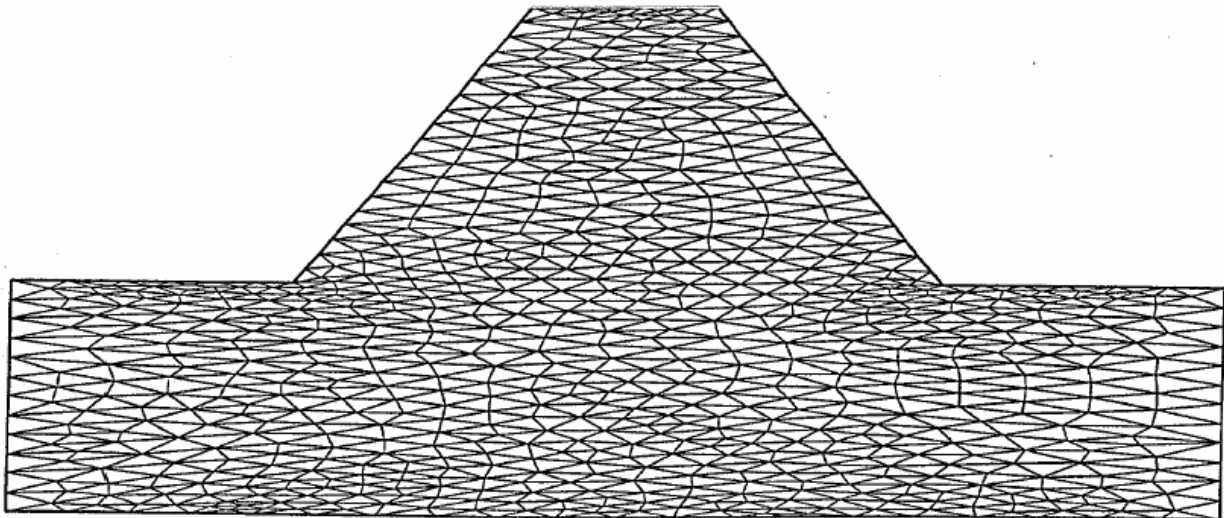
Fig. 65. Example of mesh stretching using a stretching factor of 3 in the *y*-direction.

5.6. Finite Element Mesh Statistics
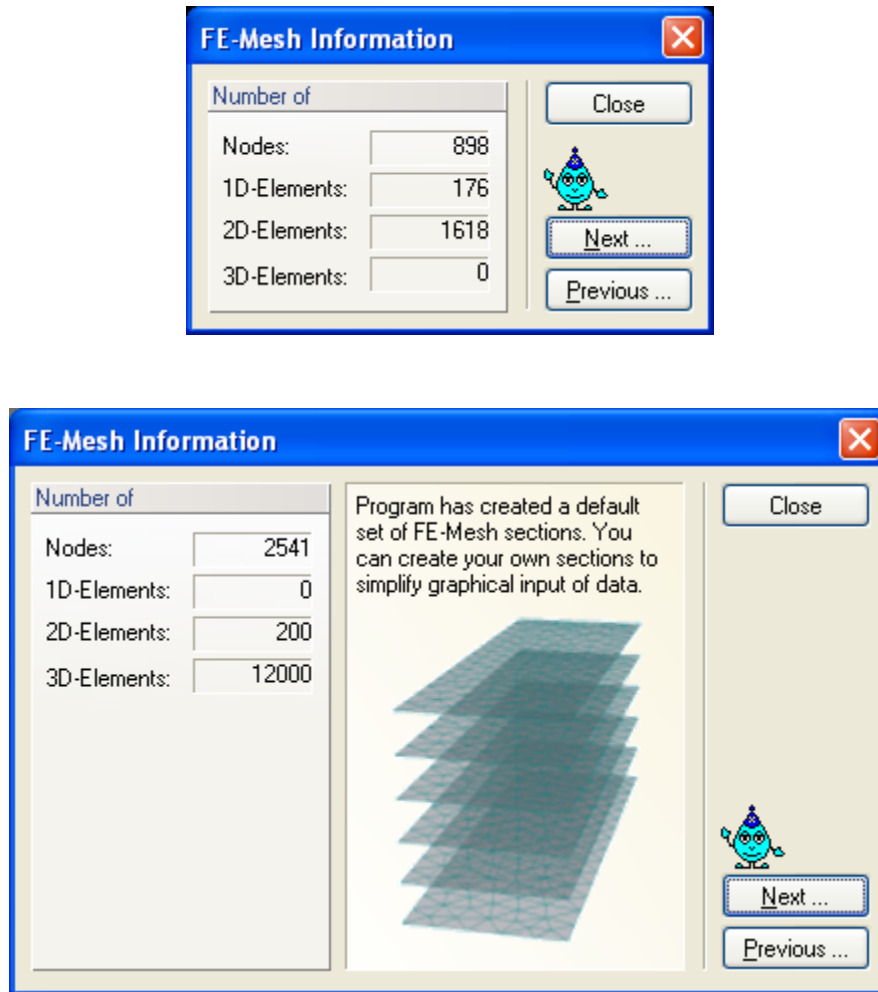
Figure 66. The FE-Mesh Information dialog window for a two-dimensional problem (top) and a three-dimensional problem (bottom).

Figure 66 shows dialog windows that provide information about the finite element mesh for two- and three-dimensional applications. For two-dimensional grids the window shows the number of finite element *Nodes*, the number of boundary *1D-Elements* (boundary edges between boundary nodes), and the number of *2D-Elements* (triangles). For three-dimensional grids the window shows again the number of finite element *Nodes* (in the entire 3D domain), the number of boundary *1D-Elements* (boundary edges between boundary nodes on the bottom plane, i.e., the 2D base surface of the transport domain), the number of *2D-Elements* (the number of triangles on the bottom plane of the domain), and the number of 3*D-Elements* (tetrahedrals) in the entire transport domain.

5.7. *Finite Element Mesh Sections*

FE-Mesh Sections are parts of the FE-Mesh used to specify input variables and to display results of calculations. By default, two-dimensional problems have only one section consisting of the entire transport domain. For three-dimensional problems one section by default is formed by the *Whole FE-Mesh* (Section D000 in Fig. 67), whereas each horizontal (vertical for some applications) layer forms one additional section (Sections D001 through D021 – *Horizontal* in Fig. 67), while the last section is made up by the vertical surface (Section D022 *Vertical* Fig. 67). For example, the bottom section can be displayed when the bottom boundary conditions are specified, while the top section can be displayed when the surface boundary conditions are provided. One can similarly display results at different depths using different horizontal sections.

Additional sections can be created using commands from the **FE-Mesh Sections** part of the **FE-Mesh** version of the **Edit Bar**. One can display any existing section (or set of sections) and modify them using the *Cut with Rectangle* command, and then create a new section using the *New Section from View* command. This new section will then appear in the list of sections in the **Section** Tab of the **Navigator Bar** and can be recalled at any time. Existing sections can be manipulated (*Display*, *Hide*, *Select*, *Unselect*, *Rename*, *Delete*, *Move Up*, and *Move Down*) using the *Edit Section* command from the **Edit Bar**, or using *Edit->Sections->Edit Sections*.
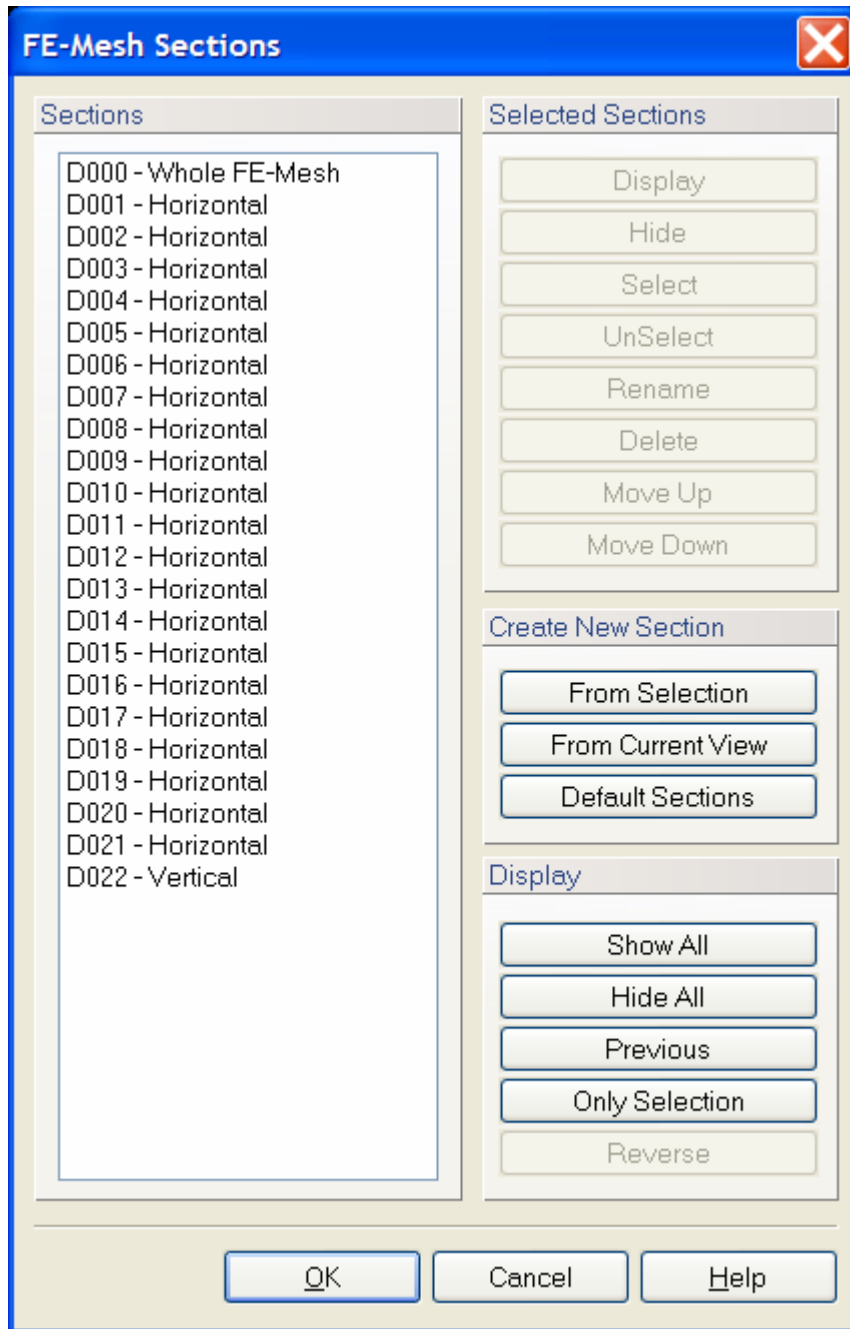
Figure 67. The FE-Mesh Sections dialog window.

6. Domain Properties, Initial and Boundary Conditions

Initial and boundary conditions for both water flow and solute (heat) transport, and the spatial distribution of other parameters characterizing the flow domain (e.g., the spatial distribution of soil materials, hydraulic scaling factors, root-water uptake parameters, and possible hydraulic anisotropy) and/or observation nodes are specified in a graphical environment with the help of a mouse. The program automatically controls the logical correspondence between the water flow and solute transport boundary conditions.

6.1. *Default Domain Properties*

For rectangular two-dimensional domains and for layered three-dimensional domains, immediately after the finite element mesh is generated, one can specify the initial **Default Domain Properties** in the dialog window shown in Figure 68. Values listed in this window are initially assigned to each horizontal layer of the transport domain, but can later be modified graphically. The following variables are involved:

Code   Code of the boundary condition (0 for no flow, -1 for constant flux, +1 for constant head, -2 for unsaturated seepage face, +2 for saturated seepage face, -3 (-7, -8, -9) for variable flux, +3 (+7, +8, +9) for variable head, -4 for atmospheric, - 5 for tile drain, -6 for free drainage)

h   Initial value of the pressure head [L]. The initial pressure head changes linearly between the first and last layer if one clicks on the command at the bottom of the dialog (*Linear interpolation of the pressure heads between the first and last layer*).

Q   Recharge flux, $[L^2T^{-1}]$ and $[L^3T^{-1}]$ for 2D and 3D applications, respectively

Mater   Material number

Roots   Root distribution

Axz   Scaling factor for the pressure head

Bxz   Scaling factor for the hydraulic conductivity

Dxz   Scaling factor for the water content

Temp   Initial temperature [K]

Conc   Initial concentration (of the equilibrium phase) $[ML^{-3}]$

Sorb   Initial concentration of the nonequilibrium phase (kinetically sorbed $[MM^{-1}]$ or of the immobile region $[ML^{-3}]$)
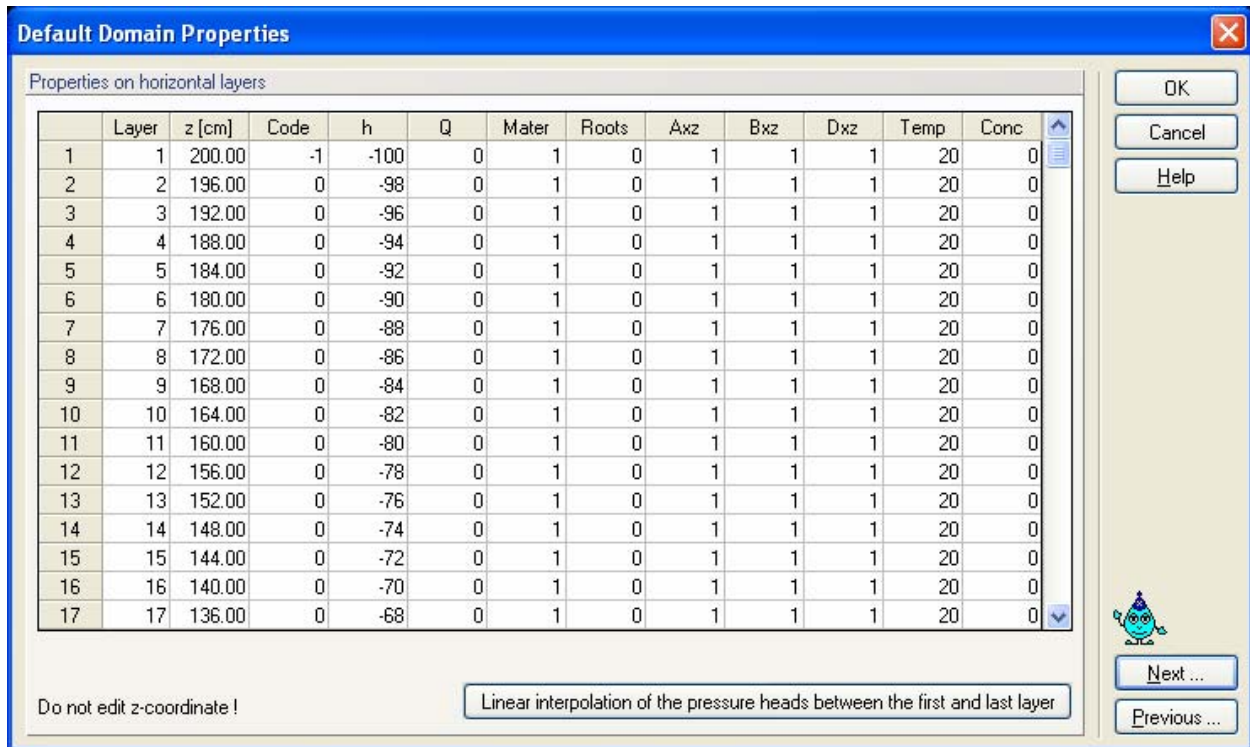
Figure 68. The Default Domain Properties dialog window.

6.2. *Initial Conditions*

After selecting nodes (graphically with the mouse) for which the initial condition is to be specified, and clicking on the command **Set Values** at the **Edit Bar** (on the right side of the view window), the **Water Flow Initial Condition** dialog window (Fig. 69) appears. Using this command one can specify the initial conditions for water flow by defining the initial spatial distribution of the pressure head or water content over the flow domain. The decision whether to use the pressure head or the water content initial distribution is made in the main module of the **Iteration Criteria** dialog window (Fig. 17). One can specify a) the same value to all selected nodes (*Same value for all nodes*), b) a distribution versus depth that is in equilibrium with the pressure head of the lowest point in the selected region (*Equilibrium from the lowest located nodal point*), and c) a *Linear distribution with depth*. When options a) or b) are selected, only one value of the pressure head needs to be specified. For option c) one needs to provide values of the pressure head (or water content) for the top and bottom of the selected domain.
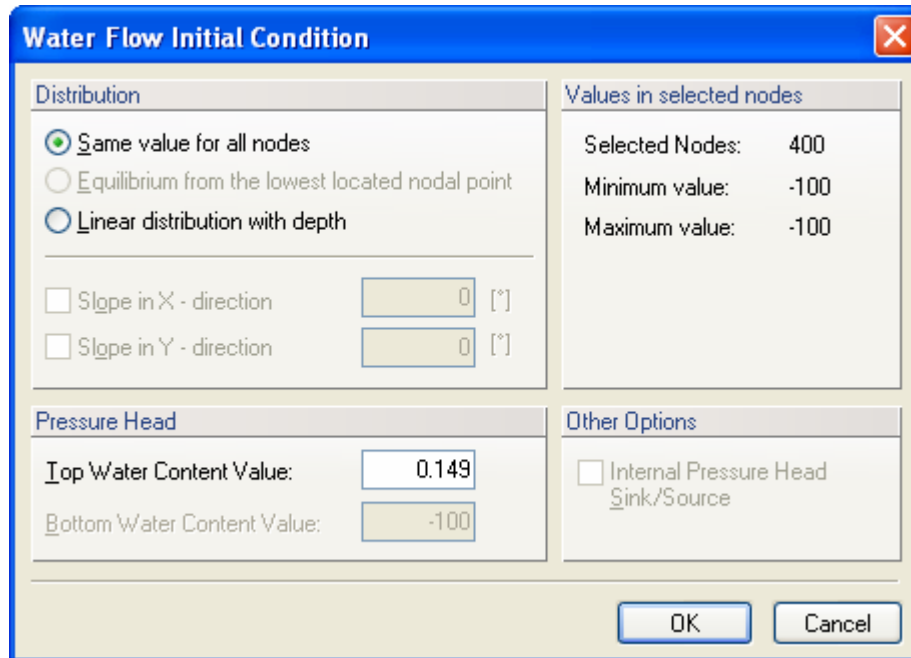
Figure 69. The Water Flow Initial Condition dialog window.

To simplify definition of the initial condition for problems that involve slopes, an option is provided to have pressure heads decrease in the *x*- and *y*- (the latter only for three dimensional problems) directions (*Slope in X- direction* and *Slope in Y- direction*).

The right top part of the dialog window (*Values in selected nodes*) in Figure 69 provides information about the selected nodes, i.e., the number of selected nodes and the minimum and maximum values of the pressure head or water content.

If a certain node is selected that is not located on the boundary, this node can be declared an *Internal Pressure Head Sink/Source*. The pressure head at that node will then be kept constant during the simulation.

Similar, but simpler, dialog windows are used to specify initial values of the temperature and the liquid and adsorbed concentrations (e.g., **Temperature Distribution** dialog window, Fig. 70). The dialog window will then again provide information about the selected nodes (*Values in selected nodes*), i.e., the number of selected nodes and their minimum and maximum temperatures (concentrations). One can specify either constant value for all selected nodes, or have the values change linearly with depth. When the box *Use top value for the entire selected region* is checked, the value in the *Top* box is assigned to all selected nodes.
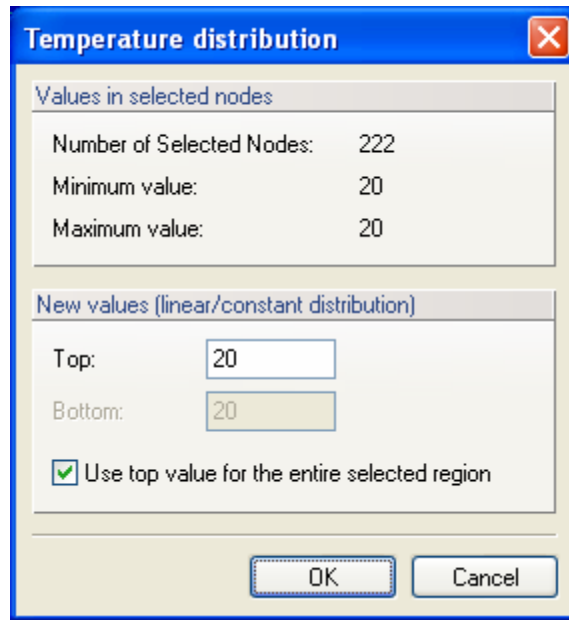
Figure 70. The Temperature distribution dialog window.

The initial condition can be imported from the results of previous calculations using the Import command (*Edit->Initial Condition->Import*, or *Insert->Initial Condition->Import*). After clicking on any of these two commands, an **Open** dialog window appears with **Files of type** preselected for HYDRUS applications (i.e.,*.h3d). One then needs to browse for the HYDRUS project from which the initial condition is to be imported. After selecting a particular project, the **Import Initial Conditions** dialog window appears (Fig. 71). This window provides information from which project the initial conditions will be imported (*Import data from Hydrus project*) and offers quantities that can be imported as initial conditions (*Select quantities to import*). Users must then also decide in the *Select Time Layer* part of the dialog if values for *The Last (Final) Time Layer* or for any intermediate time layer (using *Time Layer No.:* from the lower list box) are to be imported.
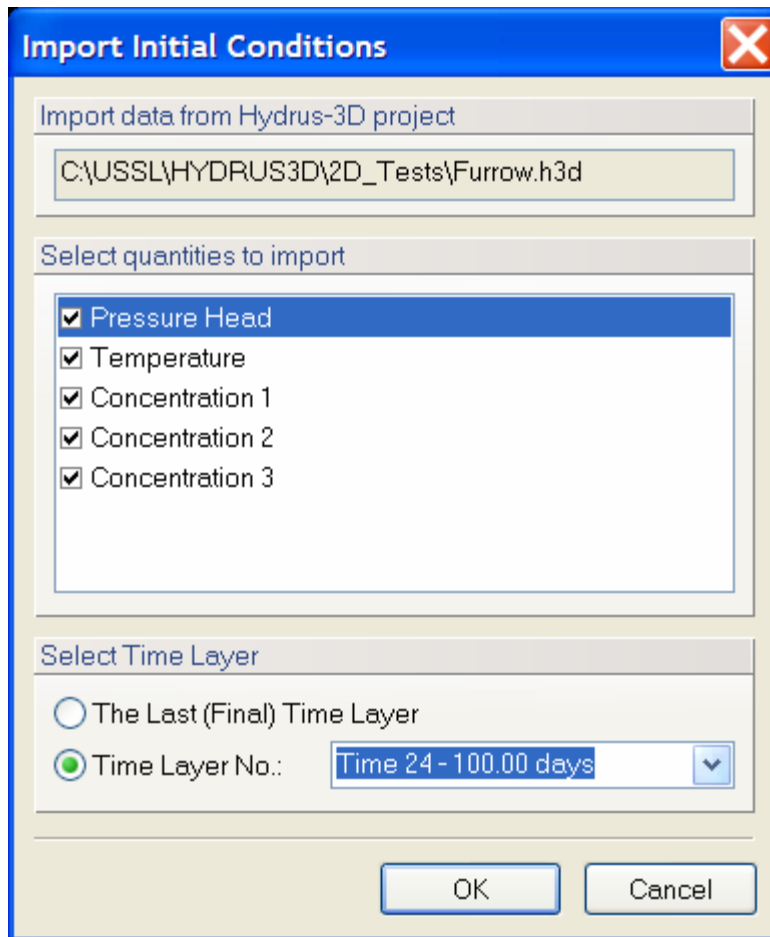
Figure 71. The Import Initial Condition dialog window.

6.3. *Boundary Conditions*

Specification of the boundary conditions is relatively straightforward. Users must first select from the **Navigator Bar** particular **Boundary Conditions** (i.e., water flow, solute transport, or heat transport) and then click on the **Edit Bar** on the particular boundary condition (e.g., constant head). They subsequently need to move the mouse to the selected position and click the left mouse button. Implementation of the boundary condition terminates with a repeated click of the left mouse button. The boundary nodes will acquire the same color as the corresponding type of the boundary condition. See the rules for specifying boundary conditions as described in Chapter 8 of the Technical Manual [*Simunek et al.* 2006].

6.4. *Domain Properties*

Other parameters characterizing the flow domain (initial condition, material distribution) are defined in a similar way. Users must first select that part of the transport domain to which he/she wants to

assign a particular value of the selected variable. It is possible to select the entire transport domain, part of it, or only individual nodes or elements. A particular part of the transport domain can be selected as follows: first move the mouse to a selected position. The beginning and end of the selection operation is framed by clicking the left mouse button. The selected area is the rectangle defined by the two mouse positions when the left mouse button was clicked. When the selection is completed, users must click the **Set values** button and specify the value of a particular variable. The given value will then be assigned to the selected area. When material numbers are to be specified, users can do this directly by clicking on the color representing a particular material (at the **Edit Bar**). All variables are assigned to nodal points, except for those defining anisotropy angles, the first and second components of anisotropy, and subregion numbers, which are all assigned to elements.

Scaling Factors can be generated for two-dimensional applications using a random generator by clicking on the command *Edit->Domain Properties->Stochastic Distribution of S.F.* that calls the **Stochastic Distribution of Scaling Factors** dialog window (Fig. 72). Each scaling factor can be generated either independently of the other scaling factors, or by assuming Miller-Miller similitude. In that case the program generates hydraulic conductivity scaling factors and automatically calculates from their values the pressure head scaling factors.
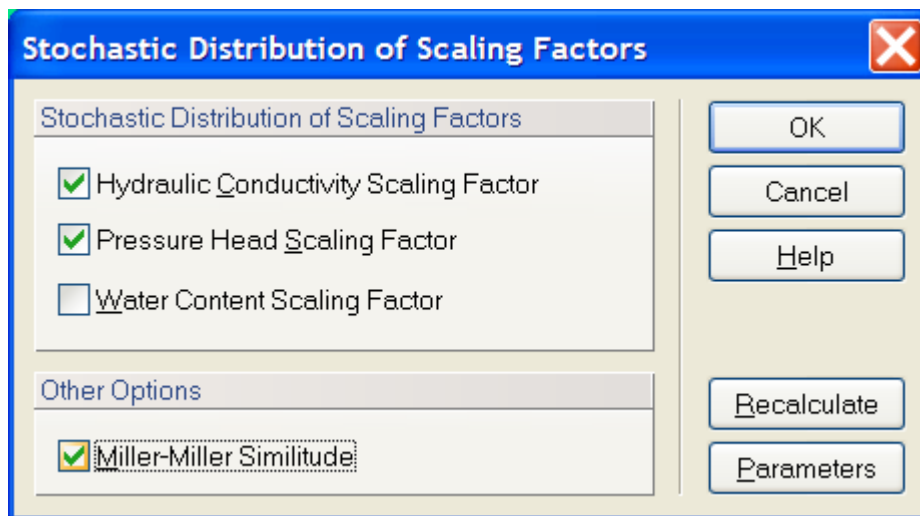
Figure 72. The Stochastic Distribution of Scaling Factors dialog window.

Parameters for random generation of the scaling factors are specified in the **Stochastic Parameters** dialog window (Fig. 73). In this dialog users need to specify whether the scaling factors are normally or log-normally distributed using the check box *Log-Normal Distribution*), and provide values for the standard deviation of a particular scaling factor, and correlation lengths in the *x*- and *z*-directions (if the scaling factors are spatially correlated).
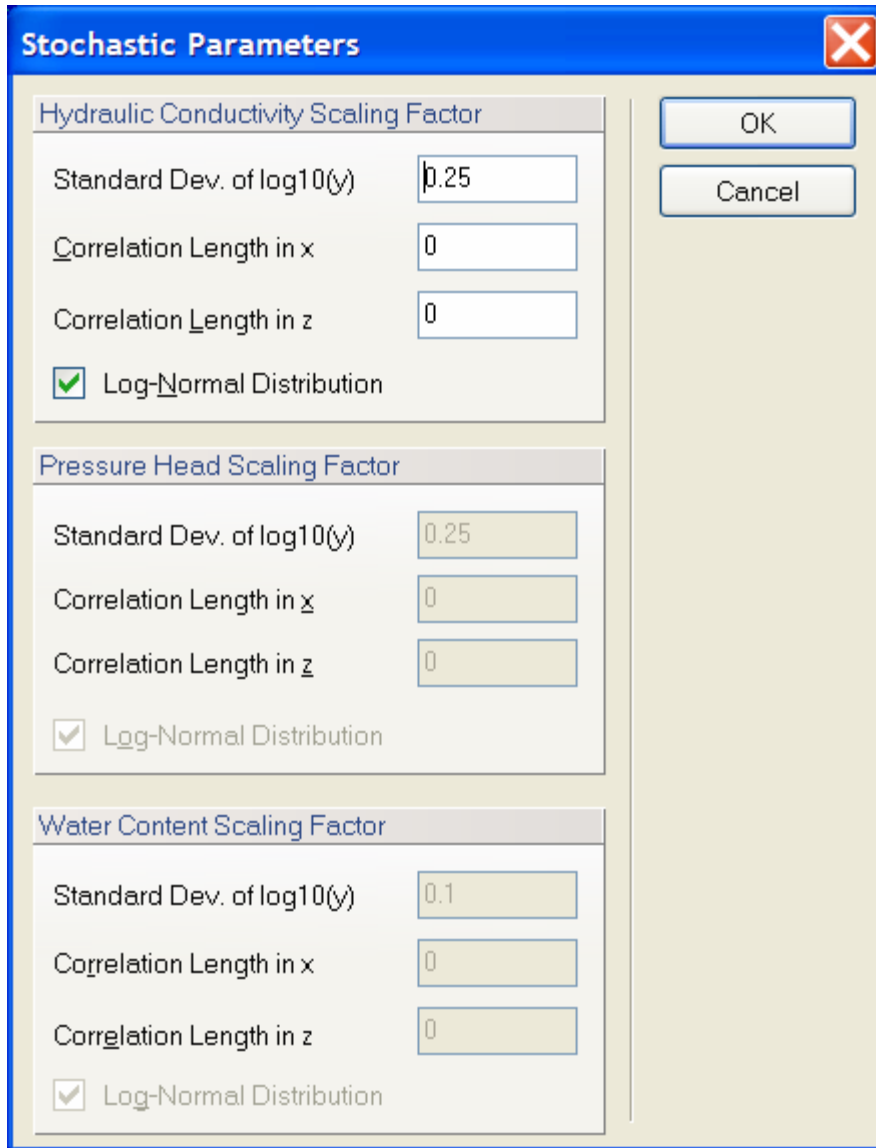
Figure 73. The Stochastic Parameters dialog window.

7. Graphical Output

Graphical output is divided into two main parts. In the first part, variables which change spatially throughout the transport domain are displayed by means of contour maps, isolines, or isobands (**Results – Graphical Display** on the **Data Tab** of the **Navigator Bar**, or *Options->Graph Type*). Additional information such as boundary fluxes and/or soil hydraulic properties are displayed using *x-y* graphs (**Results – Other Information** on the **Data Tab** of the **Navigator Bar**, or using the **Results Menu**).

7.1. *Results – Graphical Display*

Results of a simulation can be displayed by means of contour maps, isolines, isobands, color points, color edges, spectral maps, and/or velocity vectors (*Graph Type* at the **View Tab** of the **Navigator Bar** or *Options->Graph Type->Isolines*), and/or by animation using both contour and spectral maps. The number of colors in the color spectrum, as well as the numerical increment between isolines, can be selected using the **Edit Isoband Value and Color Spectra** dialog window (Fig. 75). Contour and spectral maps may be drawn for the pressure head, water content, temperature, solute concentration (in the equilibrium or nonequilibrium phase), and/or velocity. Animation of these four variables is also possible (*Flow Animation* on the *Results* version of the **Edit Bar**, or using *Results->Time Layer->Animation*). Graphs of all variables along the boundaries (*Boundary-Line Chart* on the *Results* version of the **Edit Bar**, or using *Results->Charts->Boundary Line*), as well as those along any selected cross-section (*Cross-Section Chart* on the *Results* version of the **Edit Bar** or *Results->Charts->Cross-Section*) can be readily obtained.

The entire finite element mesh, the boundary nodes, and the numbering of nodes, elements and/or edges, can be displayed also (using the **Display Options** dialog window (Fig 74), or *Options->Display Options->Edit*), together with isolines and spectral graphs. Users may zoom into a certain part of the transport domain, and can enlarge or reduce the transport domain, among other features.

Flow animation is an alternative to displaying results at one particular time. Distributions during flow animation are displayed continuously at consecutive times, thereby visualizing the flow and transport process. Note, however, that display times are defined by the print time intervals specified in the input data file. This means that the print times must be at constant intervals so that the time scale of the flow animation will not be distorted. In other words, undistorted flow animation requires that the print time intervals be constant. The speed of the flow animation depends on the hardware being used, i.e., the speed of the microprocessor and graphical card.

7.1.1. *Display Options*

The **Display Options** dialog window (Fig. 74) allows users to select
   a) how different objects from the **Category** list are to be displayed,
   b) what **colors** are to be used (colors can be redefined by clicking on the *Edit Color* command button),
   c) whether **lines** are displayed as solid, dotted, dashed or dash-dotted (*Line Type*), and what *Thickness* should be used, and
   d) the position of **numbers** for various types of *numberings*, fonts for the numbers, and whether or not numbers are *Transparent*.
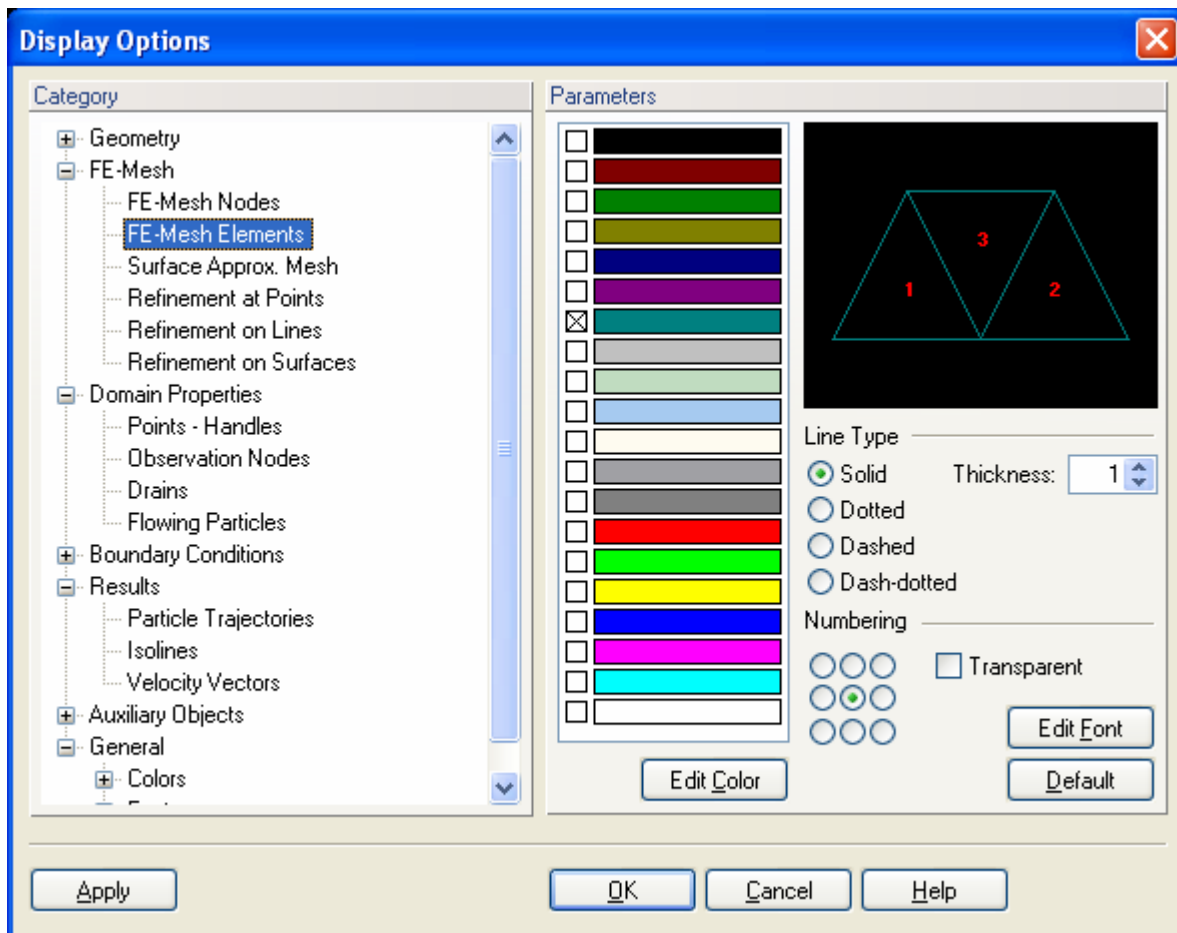
Figure 74. The Display Options dialog window.

7.1.2. *Edit Isoband Value and Color Spectra*

The **Edit Isoband Value and Color Spectra** dialog window (Fig. 75) (called by left-clicking the color scale display options of the **Results** version of the **Edit Bar**) allows users to define colors for display of isobands and color spectra, and values of particular isolines.

After double-clicking on the color panel, the **Color** dialog window (Fig. 76) appears, in which one can redefine colors to be used in displays. By default a **Standard Palette** is used. Palettes with newly defined colors can be saved under a new name and used for different purposes. The new palette can be saved locally and used only for a given application, or globally and used for all HYDRUS applications. Users hence can thus define different palettes for displaying water contents, pressure heads, concentrations, or temperatures.

Users can also define the **Isoline scales**. This can be done by simply changing numbers in the edit boxes next to the colors. If the numbers are not in an increasing or decreasing order, a warning will be issued. Newly created **Isoline scales** can be saved (again locally or globally) and used later. Users need to specify only the maximum and minimum values (the top and bottom edit boxes); after clicking the **Fill** command, the program will calculate and fill (complete) the intermediate numbers by interpolation.
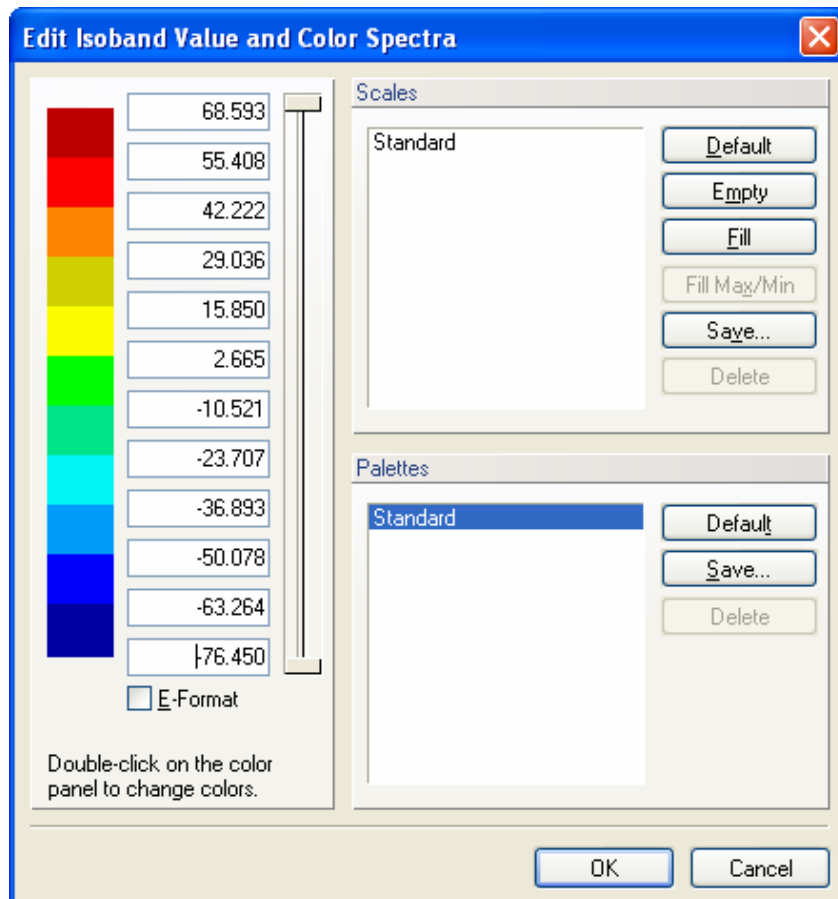
Figure 75. The Edit Isoband Value and Color Spectra dialog window.

Minimum and maximum numbers of the scale are automatically adjusted to a particular problem and for a particular variable. By default, HYDRUS searches all output time levels of a particular project for the minimum and maximum values of a particular variable, and then leaves the scale invariant in time between these two values. The scale, however, can also be automatically adjusted for each time level by specifying the minimum and maximum values for a particular variable and a particular print time when the option *Min/Max glob. in time* from the **Color Scale View Options** of the **View Tab** of the **Navigator Bar** is deselected. Similarly, the scale can be automatically adjusted for a particular layer of the FE-mesh (a section) (the minimum and maximum values for a particular variable and a particular section) when the option *Min/Max glob. in space* from the **Color Scale View Options** of the **View Tab** of the **Navigator Bar** is unselected.
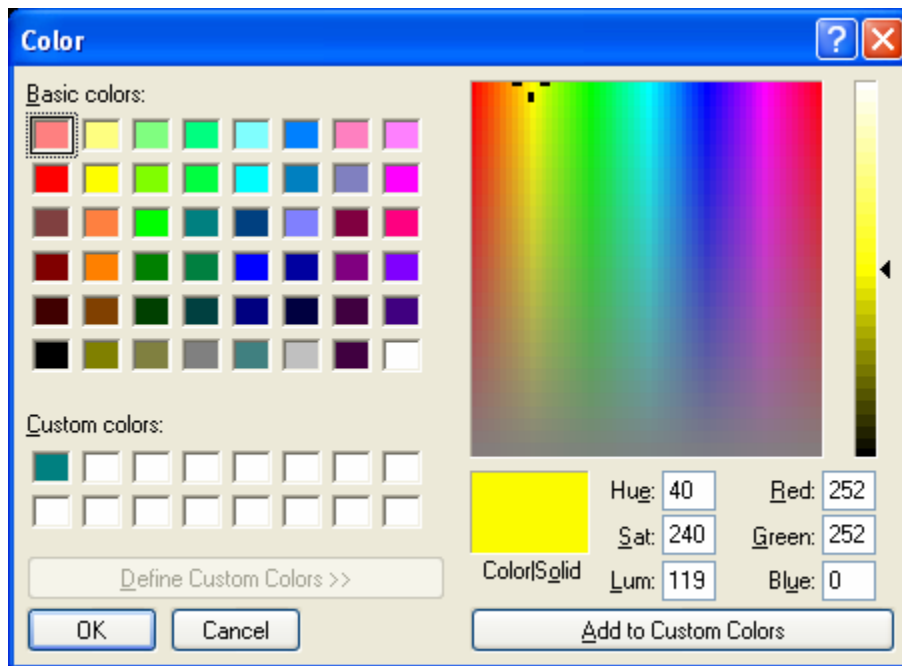
Figure 76. The Color dialog window.

7.2. *Results – Other Information*

Additional information, such as boundary fluxes and/or soil hydraulic properties can be displayed using *x-y* graphs (**Results – Other Information** on the **Data Tab** of the **Navigator Bar**, or **Results Menu** or **Results Menu**). Table 11 gives an overview of the different graph options that are possible. Additional text output is provided under the command *Mass Balance Information* on the **Data Tab** of the **Navigator Bar** (also at the **Results Menu**). This output gives the total amount of water, heat and solute inside each specified subregion, and the inflow/outflow rates to/from that subregion, together with the mean pressure head (*hMean*), mean temperature (*TMean*) and the mean concentration (*cMean*) over each subregion (see Table 11.6 of the Technical Manual). Absolute and relative errors in the water and solute mass balances are also printed to this file. Output related to the inverse problem is provided under the command *Inverse Solution Results* on the **Data Tab** of the **Navigator Bar**.

Table 11. Graph options in the HYDRUS interface.

Command	Horizontal Axis	Vertical Axis
Observation Points	Time	Pressure Head Water Content Temperature Concentration – x^+ Sorbed Concentration - x^+
Pressure Heads	Time	Atmospheric Boundary Head Root Zone Head Variable Boundary Head 1 Constant Boundary Head Seepage Face Head Drainage Boundary Head Free and Deep Drainage Boundary Head Variable Boundary Head 2 Variable Boundary Head 3 Variable Boundary Head 4 All Boundaries
Water Boundary Fluxes	Time	Potential Atmospheric Flux Potential Root Water Uptake Rate Actual Atmospheric Flux Actual Root Water Uptake Rate Variable Boundary Flux 1 Constant Boundary Flux Seepage Face Flux Drainage Boundary Flux Free and Deep Drainage Boundary Flux Variable Boundary Flux 2 Variable Boundary Flux 3 Variable Boundary Flux 4 All Atmospheric Fluxes All non-Atmospheric Fluxes
Cumulative Water Boundary Fluxes	Time	Potential Atmospheric Flux

		Potential Root Water Uptake Rate
		Actual Atmospheric Flux
		Actual Root Water Uptake Rate
		Variable Boundary Flux 1
		Constant Boundary Flux
		Seepage Face Flux
		Drainage Boundary Flux
		Free and Deep Drainage Boundary Flux
		Variable Boundary Flux 2
		Variable Boundary Flux 3
		Variable Boundary Flux 4
		All Boundaries Fluxes
Solute Fluxes[+]	Time	Cumulative Zero-Order Reaction
		Cumulative First-Order Reaction
		Cumulative Root Solute Uptake
		Cumulative Non-Equil. Mass Transfer
		Cumulative Constant Boundary Solute Flux
		Cumulative Seepage Face Solute Flux
		Cumulative Variable Boundary 1 Solute Flux
		Cumulative Atmospheric Solute Flux
		Cumulative Drain Boundary Solute Flux
		Cum. Free/Deep Drainage Bound. Solute Flux
		Cumulative Variable Boundary 2 Solute Flux
		Cumulative Variable Boundary 3 Solute Flux
		Cumulative Variable Boundary 4 Solute Flux
		Constant Boundary Flux
		Seepage Face Flux
		Variable Boundary Flux 1
		Actual Atmospheric Flux
		Drain Boundary Flux
		Free and Deep Drainage Boundary Flux
		Variable Boundary Flux 2
		Variable Boundary Flux 3
		Variable Boundary Flux 4
		All Solute Cumulative Fluxes
		All Solute Fluxes
Soil Hydraulic Properties	Pressure Head	Water Content
	Log Pressure Head	Soil Water Capacity
	Water Content	Hydraulic Conductivity
		Log Hydraulic Conductivity
		Effective Water Content
		Pressure Head
		Log Pressure Head
Run-Time Information	Time Level	Time Step
	Time	Number of Iterations
		Cumulative Number of Iterations
		Peclet Number
		Courant Number
		Number of Solute Iterations

[+] This graph is given for each solute

7.2.1. *Convert to ASCII*

The output files H.OUT, TH.OUT, CONCx.OUT, SORBx.OUT, TEMP.OUT, and V.OUT provide binary output of specific variables. The user interface can convert these binary files into the ASCII files H.TXT, TH.TXT, CONCx.TXT, SORBx.TXT, TEMP.TXT, and V.TXT using the **Convert to ASCII** dialog window (Fig. 77) (**Results – Other Information** on the **Data Tab** of the **Navigator Bar**, or *Results->Convert Output to ASCI*).
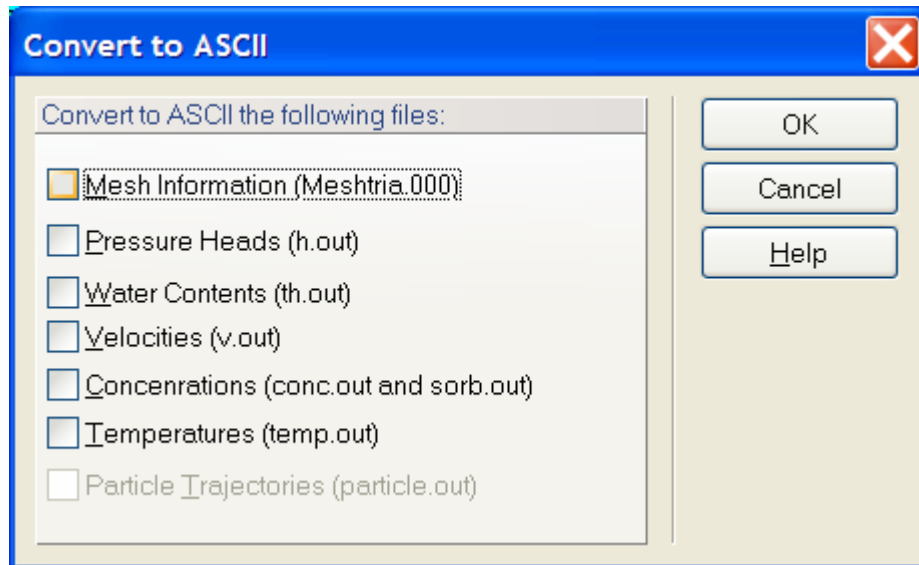
Figure 77. The Convert to ASCII dialog window.

8. Graphical User Interface Components

8.1. *View Window*

8.1.1. *Scene and Viewing Commands*

We will use here the term '**Scene**" for the content of the **View** window. Ffour types of commands are available to change the display of the scene in the **View** window.

a) Commands to define a required **Display of the Scene**.
Detailed information about particular commands is given in Section 8.4. In addition to those commands, it is always possible to adjust the **Display of the Scene** using the "mouse scroll wheel" as follows:
- Simultaneously holding the mouse scroll wheel and moving the mouse will move the **Scene** in the same direction as the mouse, i.e., to the left, right, up, or down.
- Simultaneously holding the **Ctrl** keyboard button and the mouse scroll wheel, while moving the mouse, rotates the **Scene** around the center of the displayed object (available only for three-dimensional objects).
- Simultaneously holding the **Shift** keyboard button and the mouse scroll wheel, while moving the mouse, leads to zooming in or out of the **Scene** from the center of the **View** window.
- Rotating the scroll wheel leads to zooming in or out of the **Object** from the point to which the mouse is directed.

b) Commands to define the **Content of the Scene**.
In every view one can independently specify what is to be displayed, e.g., a variable, the type of graph, or the numbering of objects. All possible options related to the **Content of the Scene** are located on the **Navigator Bar** of the **View Tab** (Section 8.2).

c) Displaying and/or hiding parts of complicated objects.
One often needs to display only some part of a complicated object while hiding the rest. For this purpose one can use commands related to **Sections** (i.e., parts of the computational domain or FE-Mesh). Detailed information is given in Chapter 8.1.8 on **Sections**.

d) Colors, fonts, and type of lines.
One can define colors, the style and thickness of lines, fonts for numbering, and other displaying options for almost all displayed/used objects. Separate default sets exist for the display for the screen and the printer. Users can create and save additional combinations of display options. Detailed information is provided in Chapter 7.1.1 on **Display Options**.

8.1.2. *Grid and Work Plane*

The **Grid and Work Plane** dialog window (Fig. 78) allows users to
a) select a **Work Plane** (i.e., a plane in which users can specify various boundary objects, initial and boundary conditions, or other information),
b) define the **Origin** of the coordinate system, and
c) define the *Alignment Grid*. The **Grid** is defined by its *Origin*, the type (*Grid Type*) of coordinate system involved (either *Cartesian* or *Polar*), and the *Grid Spacing*. The **Grid** can also be rotated to facilitate work, for example when defining the domain for a hill slope problem.
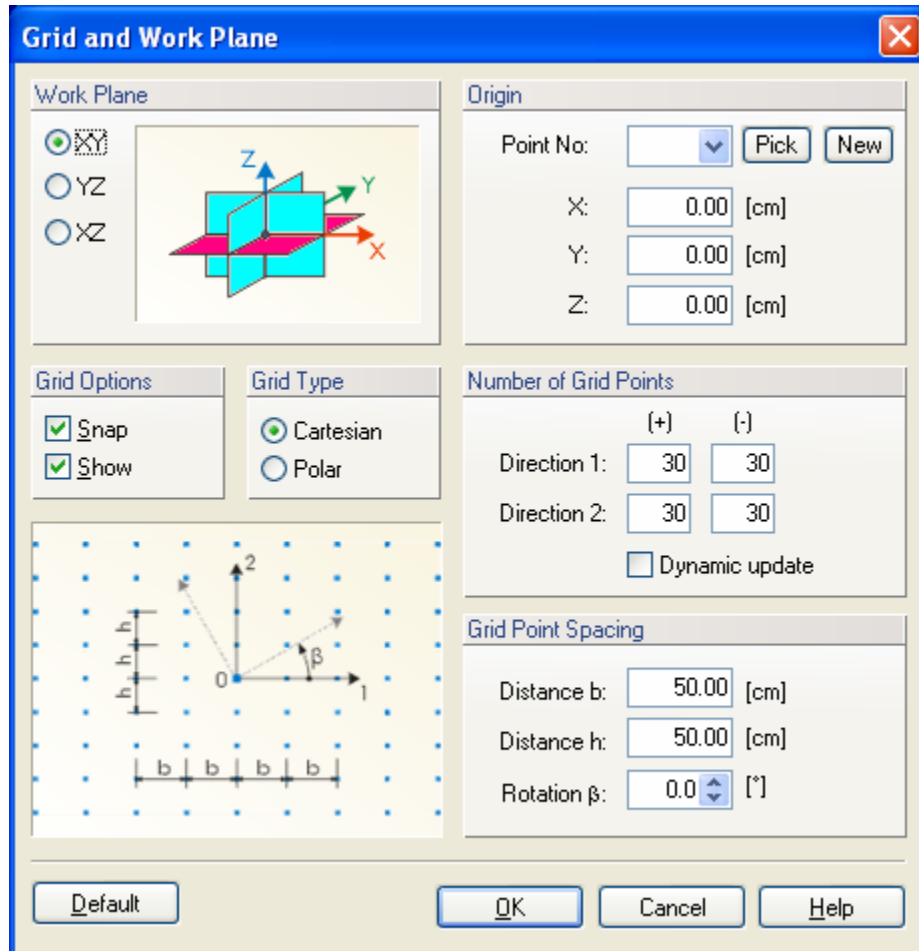
Figure 78. The Grid and Work Plane dialog window.

8.1.3. *Stretching Factors*

In many applications one direction of the transport domain dominates the other direction (or the other two directions). To facilitate work in the graphical environment, HYDRUS allows stretching of the domain in one (or two) direction(s) using **Stretching Factors**. This is done using the **View Stretching Factors** dialog window (Fig. 79) that is called with the command *View->View Stretching*.
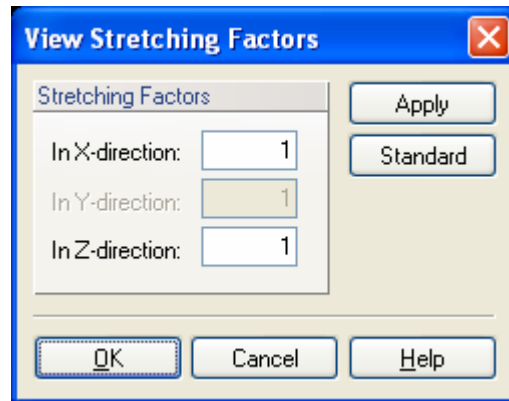
Figure 79. The View Stretching Factors dialog window.

8.1.4. *Rendering Model*

Rendering serves to rapidly switching between displays of surfaces and solids. One can select the **Wire** and **Full Rendering** options. An advantage of **Wire Rendering** is mainly the speed of the display, while **Full Rendering** provides a more realistic display of three-dimensional objects.
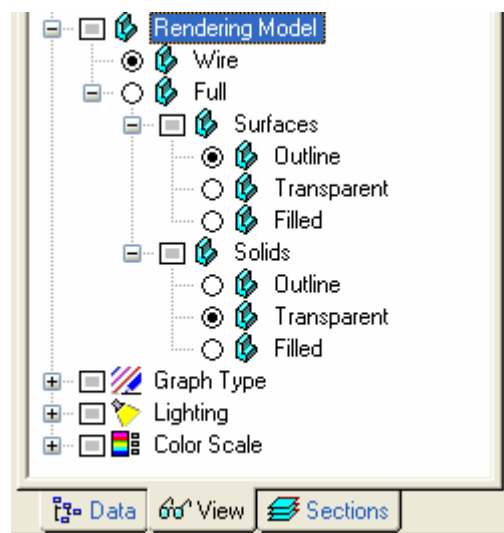
Figure 80. The Rendering part of the View Tab of the Navigator Bar.

8.1.5. *Selection and Edit Commands*

The **selection** of graphical objects is based on standards used in MS Office. An object is selected by clicking the left mouse button; a multiple selection is made by simultaneously holding the Shift button. A selection can be made using a rectangle (by clicking with the left mouse button outside of objects and holding the left mouse button while moving the mouse) or rhomboid (see Chapter 8.4. on **Toolbars** and **Tools**).

When the cursor (mouse) is moved above an object, information about that object appears on the status bar and the object is temporarily highlighted; this process is referred to as **pre-selection**. Apart from selecting objects graphically, they can also be selected using their **Indices** or by means of **Sections**.

Double-clicking on selected objects or simultaneous holding the Alt+Enter buttons recalls dialogs for editing properties of particular objects. Most dialogs support **multiple editing**, i.e., if edit boxes with different values remain empty, the original values will not change. This feature allows for example *Z*-coordinates of multiple selected points to be changed, while leaving the *X*- and *Y*-coordinates unchanged.

When different objects are selected simultaneously (e.g., points and curves), double clicking causes a dialog window to appear from which objects can be selected for editing.

8.1.6. *Pop-up Menus*

Context-sensitive menus with useful commands for a particular object can be called from the **View** window when clicking the right mouse button. While the commands are accessible also from the main menu, right-clicking the mouse is much faster. Menus for multiple selections that may contain different types of objects operate in the same way. When the right mouse button is clicked a default menu will appear when no object is close to the cursor.

When one clicks with the right mouse button on the view window, the pop-up menu of Figure 81 will appear. This menu will allow users to
    a) select different views (*View*), i.e., Isometric, in *X*-direction, in *Y*-direction, in *Z*-direction, Reverse *X*-direction, Reverse *Y*-direction, Reverse *Z*-direction, or a perspective view.
    b) select *Numbering* (turns on and off the display of numbering for objects selected under *Numbering Options* on the **Navigator Bar**).
    c) go quickly to various *Numbering Options* (this command will bring the **Navigator Bar** to the View Tab on Numbering).
    d) switch *Full Rendering/Wire Rendering* (see 8.1.4).
    e) start the *Autorotate* function that will rotate the transport domain in the View window
    f) call the *Show Work Plane*, i.e., to show the axis and the origin of the Work Plane
    g) call the **Set Grid and Work Plane** dialog window (Fig. 78) (*Set Grid and Work Plane*)
    h) select the *Coordinate System* (call the **List of Available Coordinate Systems** dialog window), and
    i) call the **Display Options** dialog window (Fig. 74)
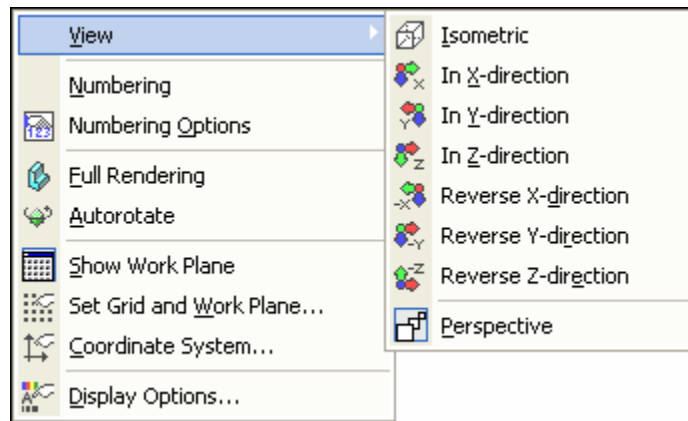
Figure 81. The Pop-up Menu from the View window.

8.1.7. *Drag and Drop*

By clicking and holding the left mouse button on selected objects in the **View** window, the **Drag and Drop** operation is started. This operation allows selected objects to be moved to a new location. Simultaneous holding the Ctrl keyboard button leads to the creation of a copy of the selected object. The copy can be moved into a different view, or in a different open project. The operation terminates by holding the Esc button or by clicking the right mouse button. **Drag and Drop** can be used for most geometric objects, but also forh auxiliary objects such as comments, labels, and bitmaps.

Holding the Shift button during the **Drag and Drop** operation leads to movement of selected objects in a direction perpendicular to the current **Working Plane** (available only for three-dimensional objects).

8.1.8. *Sections*

Sections serve do divide complex objects (models) into simpler parts. Only the simpler parts are then displayed in the **View** window while the remaining parts are hidden. Two types of sections exists: those for geometric objects and those for the FE-Mesh. New sections for both types can be created and named, a list of which is displayed on the **Section Tab** of the **Navigator Bar**. Clicking on the **Section** of the **Navigator Bar** causes the **Section** to be displayed in the **View** window. Multiple **Sections** can be displayed simultaneously by holding the "Shift" button during the selection. Undesired (to be displayed) parts can be cut off and hidden from the **View** window using the **Cut with Rectangle** command. This leads to a temporary section that is remembered by the program and renewed after the project is closed and reopened. New **sections** can be named and listed in the **Navigator Bar** by using commands to create new sections. The simplest command is the **New Section from View** command, which creates a newly named **section** from currently displayed objects in the **View** window. The **New Section from Selection** command creates a newly named section from currently selected objects or the FE-Mesh. The **Display All** and **Display whole FE-mesh** commands causes the entire computational domain or FE-Mesh to be displayed.

8.2. *Navigator Bars*

The **Navigator Bar** (Fig. 82) is by default located on the right side of the HYDRUS main window. A user can, however, move the **Navigator bar** to other positions. The **Navigator Bar** has three Tabs:

a) A **Data Tab** to allow quick access to all input and output data. Input data include *Domain Geometry*, *Flow Parameters*, *FE-Mesh*, *Domain Properties*, *Initial* and *Boundary Conditions*, and *Auxiliary Objects*. Output data include various *Results*. Data are organized in a tree-like structure. Figure 82 shows *Data* with expanded *Domain Properties* on the left and *Results* in the middle.
b) A **View Tab** (on the right of Figure 82) to specify what and how information will be displayed in the *View window*, and
c) A **Sections Tab** to show various Sections (not shown here and defined elsewhere).

View Options on the **View Tab** of the **Navigator Bar** at first mirror more or less the **Project Data**, i.e., **View Options** exists for *Domain Geometry*, *FE-Mesh, Domain Properties*, *Initial* and *Boundary Conditions*, and *Results*. Users can then select which objects (*Domain Geometry objects*, *Domain Properties objects*, *FE-Mesh objects*, *such as nodes, edges, triangles,* and *tetrahedrals*) are to be numbered (*Numbering*), display *Auxiliary Objects*, and initiate *Rendering* (*Outline, transparent, filled*). Users can also select the *Graph Type* (*Isolines*, *Contours* with or without *Separation Lines*, and *Color Edges*, *Color Points*, and *Velocity Vectors*) and *Color Scale* (either with or without *Min/Max glob. in time* or *Min/Max glob. in space*). Finally, *Lighting* can be turned on or off; the location of light sources can be shown (*Show Light Sources*), and the location of lights can be selected (*Light Switches*).
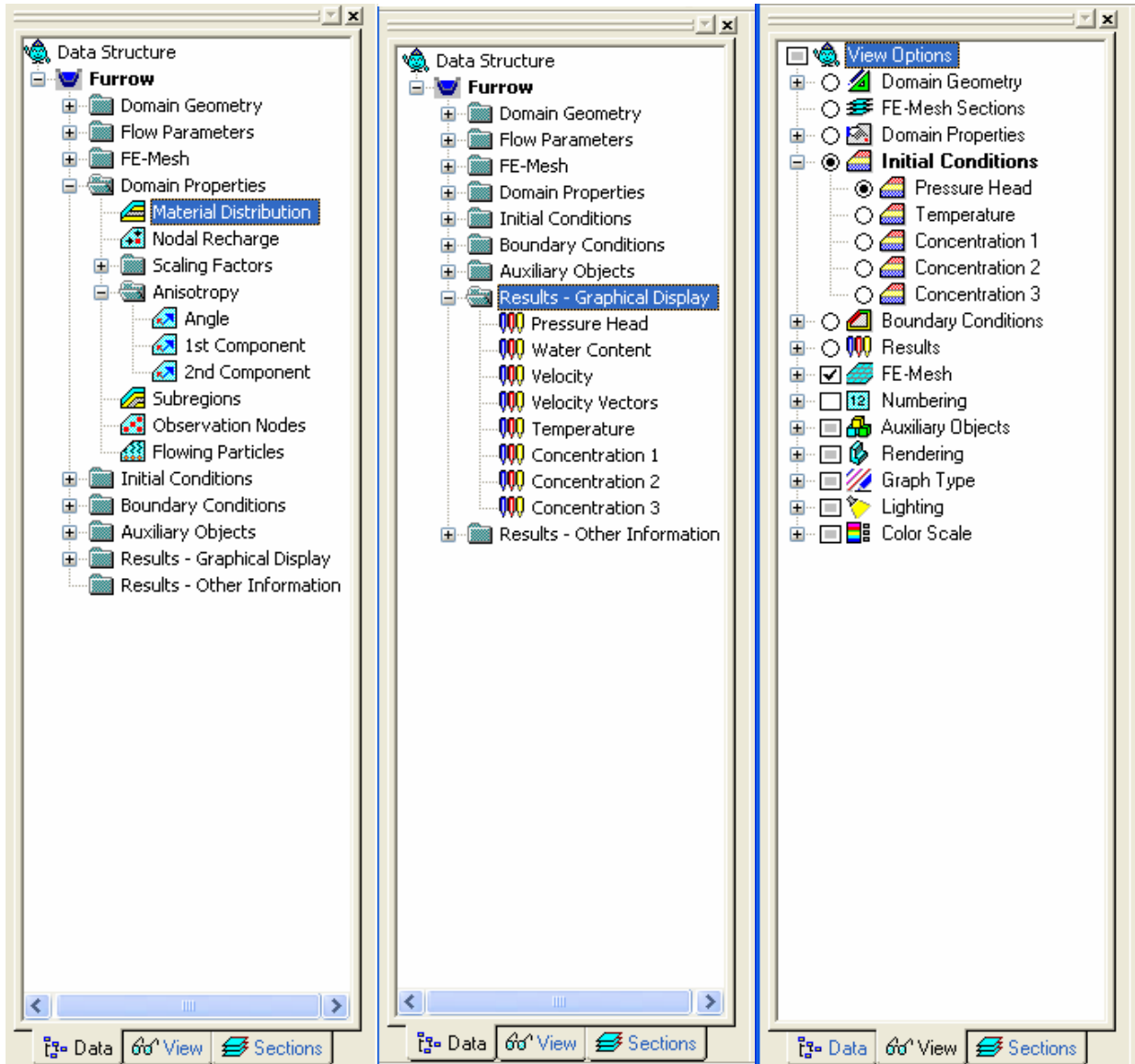
Figure 82. Selected Navigator Bars (Data Tabs on the left and in the middle, the View Tab on the right).

8.3. *Edit Bars*

The **Edit Bar** is by default located on the right side of the HYDRUS main window. A user can, however, move the **Edit Bar** to other positions. The **Edit Bar** is very dynamic since it changes depending upon the process being carried out. Figure 83 shows the **Edit Bars** for different processes, i.e., for the Material Distribution in the *Domain Properties*, the Water Flow *Boundary Conditions*, the Pressure Head *Initial Conditions*, and the Water Content *Results*.
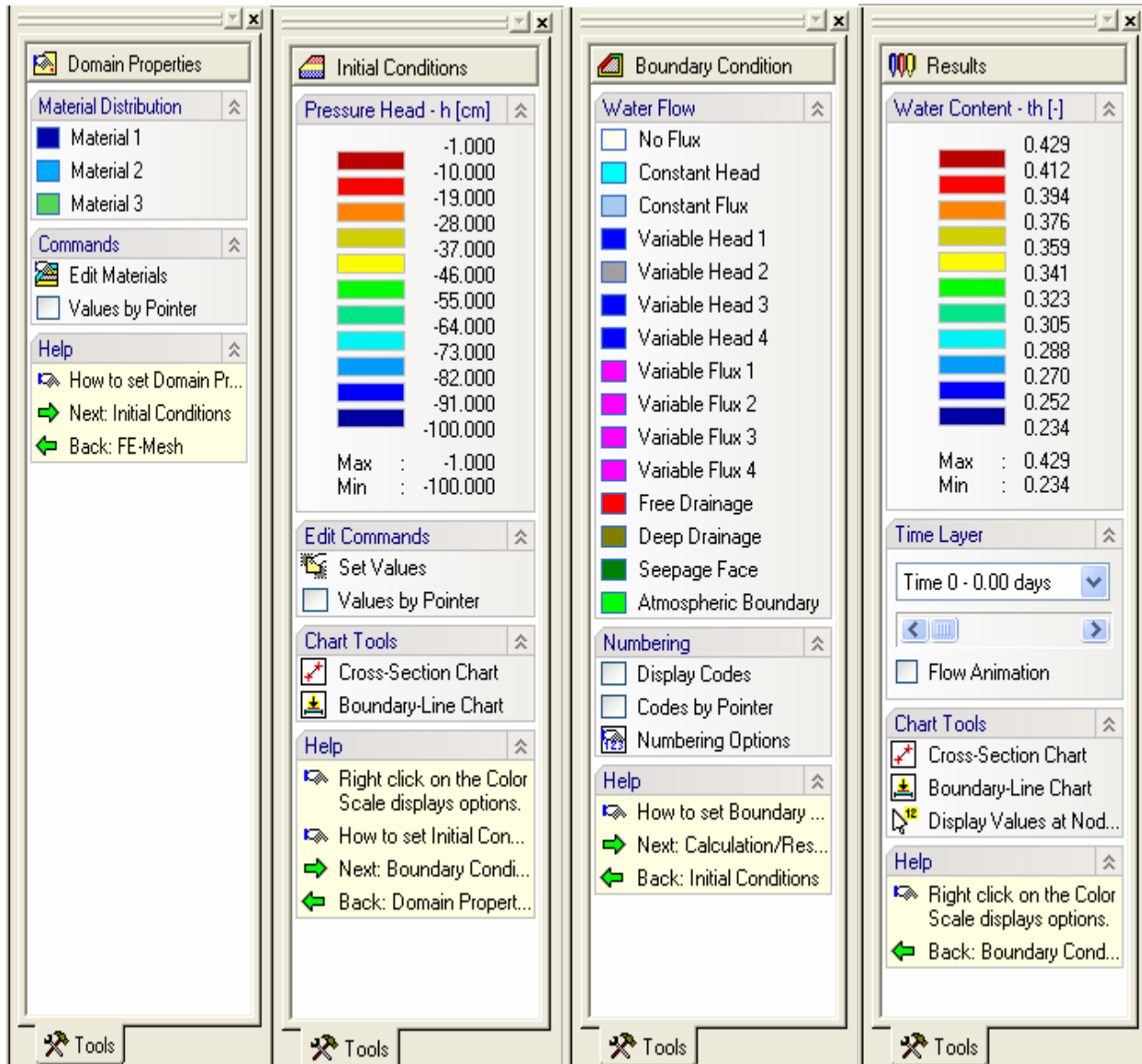
Figure 83. Selected Edit Bars (from left to right) for Material Distribution in Domain Properties, Water Flow Boundary Conditions, Pressure Head Initial Conditions, and Water Content Results.

As an example, the **Edit Bar** for **Domain Properties** and **Material Distribution** displays materials that can be assigned to selected nodes (in this case Materials 1, 2, and 3), the Command **Edit Materials** (which calls the **Water Flow Parameters** dialog window, Fig. 19), the command **Values by Pointer** (which displays the material number for a node closest to the cursor), and **Help**. The **Help** part of the **Edit Bar** usually contains help on a particular process (*How to …*), and the direction of the data inputting process (next or backwards).

The **Edit Bar** for **Boundary Conditions** and **Water Flow** displays the various boundary conditions that can be specified along boundaries of the transport domain, as well the command **Display Codes** (which displays boundary codes for all boundary nodes in the View window), the command **Codes by Pointer** (which displays boundary codes only for the node closest to the cursor), **Numbering Options** (which changes the **Navigator Bar** to the **View** Tab and *Numbering* option), and **Help** (which is similar as discussed above).

The **Edit Bar** for **Initial Conditions** and **Pressure Head - h** displays a color spectrum that is used to draw the initial conditions, and lists the minimum and maximum values that are used in the entire domain. This **Edit Bar** also includes:
   a) **Edit Commands**: **Set Values** and **Values by Pointer**. When nodes for which the initial conditions are to be specified are already selected, then the **Set Values** command calls the **Water Flow Initial Condition** dialog window (Fig. 69). When no nodes are selected, then clicking on the **Set Values** command causes a square cursor to appear, which may be used to select particular nodes, after which the **Water Flow Initial Condition** dialog window appears. The command **Values by Pointer** again displays the initial pressure head of the node closest to the cursor.
   b) Two **Chart Tools** commands: The **Cross-Section Chart** and the **Boundary-Line Chart**. The **Cross-Section Chart** command allows users to display a particular variable between any two points of the transport domain. The **Boundary-Line Chart** command allows users to display a particular variable between any two points on the boundary of the transport domain, or along any line that is drawn along edges of finite elements within the transport domain. This line hence does not have to be straight, but can turn in any direction along finite element edges.
   c) The **Help** commands as before.

The **Edit Bar** for **Results** and **Water Content** displays, similarly as for the initial conditions, a spectrum that is used to draw results, and the minimum and maximum values for the entire domain. This bar further includes
   a) A **Time Layer** command to specify which time layers (corresponding to print times specified in the **Output Information** dialog window, Fig. 16) are to be displayed. Time layers can be chosen either from the list box, or from a scroll bar. It is also possible to perform animation of results by clicking on the **Flow Animation** check box.
   b) Two **Chart Tools** commands (**Cross-Section Chart** and **Boundary-Line Chart**), which have the same purpose as above for the initial condition. The last command (i.e., **Display Values at Nodes**) again causes the value of a particular variable (e.g., water content) of the node closest to the cursor do be displayed.

c) The **Help** command. One useful feature of the **Help** command here is the *Right click on the Color Scale displays options* (a right-click displays the **Edit Isoband Value and Color Spectra** dialog window, Fig. 75). Right-clicking results in the display of the pop-up menu (Fig. 84) that allows users to choose different display options, such as *Color Smoothing*, *Isolines*, and *Isobands*. These options are defined in more detail below.
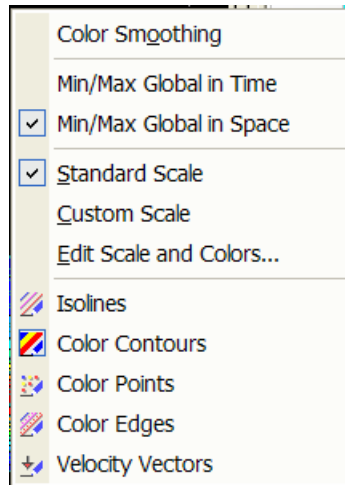
Figure 84. The Color Scale Display Options menu.

Color Smoothing	Colors are by default constant between isolines, but change abruptly at the isoline. Colors change gradually when this option is checked.
Min/Max Global in Time	The minimum and maximum values for the color spectrum are selected based on minimum and maximum values of a certain variable during the entire simulation.
Min/Max Global in Space	The minimum and maximum values for the color spectrum are selected based on minimum and maximum values of a certain variable in the entire transport domain (even when only part of the domain is displayed, e.g., one horizontal layer).
Standard Scale	Users can select between a standard or user-defined scale.
Custom Scale	Users can select between a standard or user-defined scale.
Edit Scale and Colors	Calls the **Edit Isoband Value and Color Spectra** dialog window (Fig. 75).
Isolines	Displays a selected variable using isolines.
Color Contours	Displays a selected variable using isobands (color spectrum).
Color Points	Displays a selected variable using color points.
Color Edges	Displays a selected variable using color edges.
Velocity Vectors	Displays velocities using velocity vectors.

Figure 85 shows two more **Edit Bars**, i.e., those for *Domain Geometry* and *FE-Mesh*. The **Insert Object** of the **Domain Geometry Edit Bar** allows users to **Insert Objects** with which a transport domain is defined (i.e., points, lines, arcs, circles, splines, surfaces), as well as auxiliary objects such as dimensions and comments. The **Edit Objects** part allows objects to be edited using various actions, such as *Move*, *Copy*, *Rotate*, and *Mirror*. This **Edit Bar** also provides **Help** on how to *Edit Domain Geometry*, and to *Check Domain Definition.*

The **FE-Mesh Edit Bar** allows quick access to various commands needed for editing and generating the finite element mesh (**Edit FE-Mesh**), such as the *FE-Mesh Generator* (Fig. 56), *FE-Mesh Parameters* (Figs. 59 through 62), *Insert Mesh Refinement* (Fig. 63), *Delete All Refinements*, *Generate FE-Mesh*, *Delete FE-Mesh*, and *FE-Mesh Information* (Fig. 66). The **FE-Mesh Edit Bar** also allows users to generate the finite element mesh step by step (**FE-Mesh Advanced**), i.e., using individual steps such as *Fundamental Triangulation*, *Mesh Refinement*, *Delaunay Retriangulation*, *Convex Retriangulation*, and *Mesh Smoothing*. This **Edit Bar** additionally provides tools to work with **FE-Mesh Sections** and **Help**.
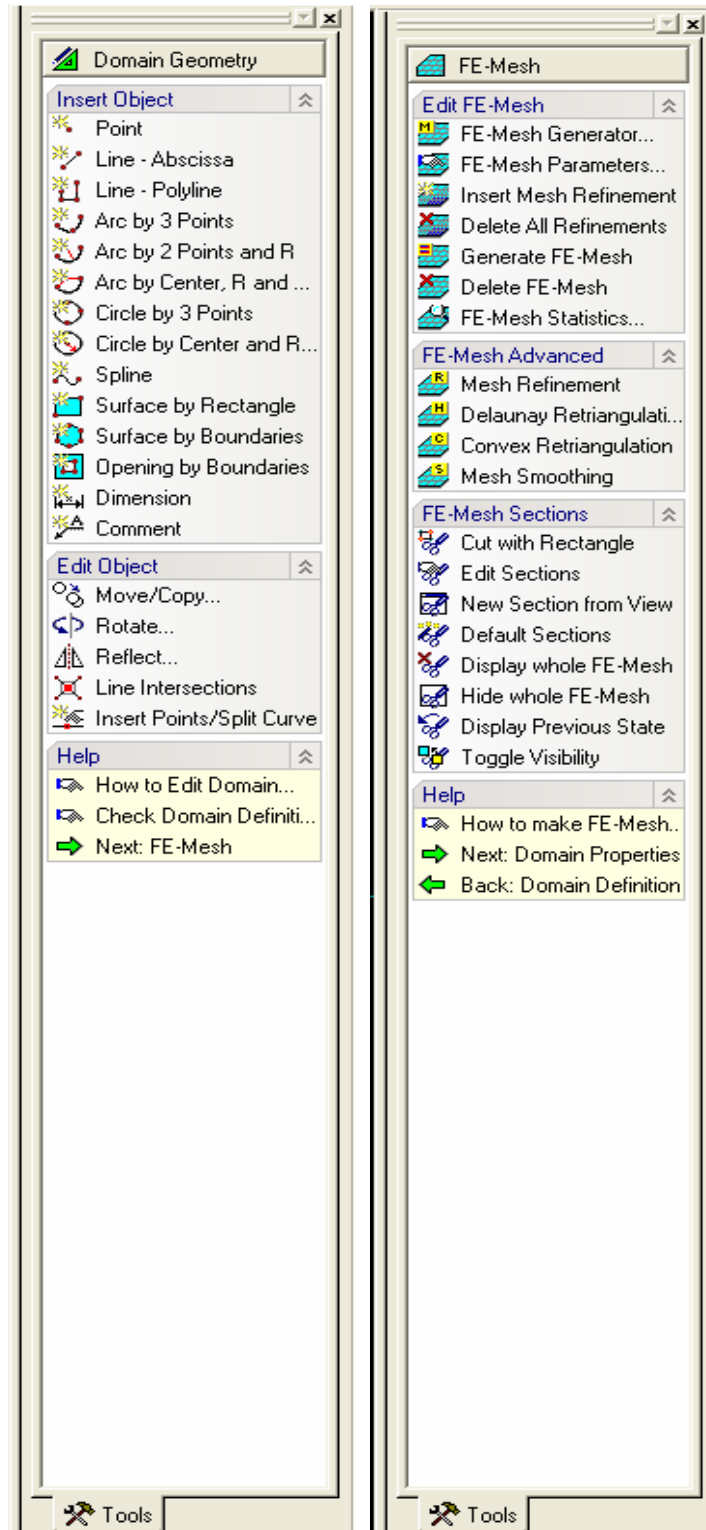
Figure 85. Selected Edit Bars (for Domain Geometry and FE-Mesh).

8.4. *Toolbars*

Users can use various **toolbars** that allow an easy access to the most frequently used commands. These commands are grouped into five toolbars that can be displayed using the command *View->Toolbars* (Fig. 86). Users can also create their own toolbars and customize them in various ways using the **Customize** dialog window (Fig. 87).
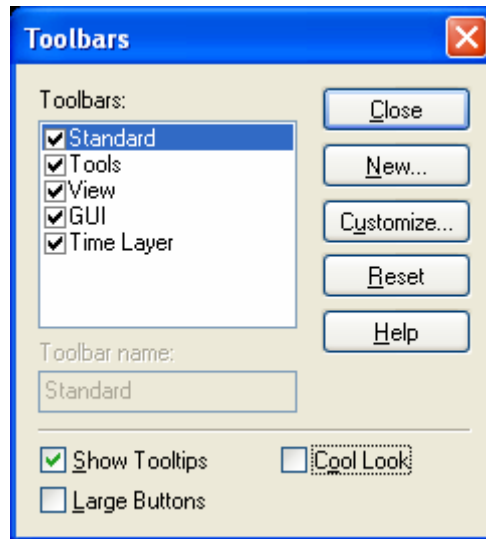
Figure 86. The Toolbars dialog window.
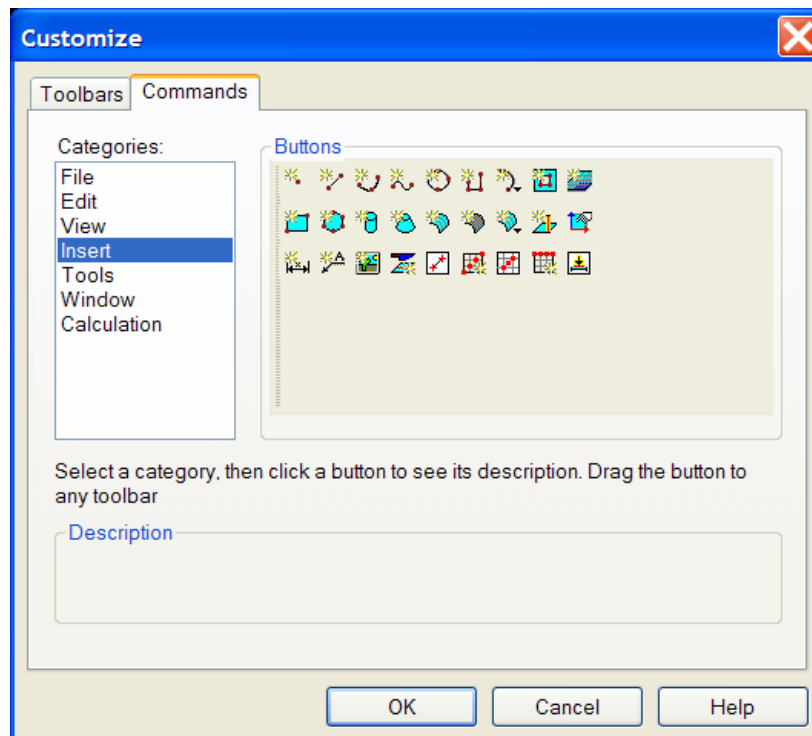
Figure 87. The Customize Toolbars dialog window.

131

The default toolbars (i.e., **Standard Toolbar**, **Tools Toolbar**, **View Toolbar**, **GUI Toolbar**, and **Time Layer Toolbar**) are briefly summarized below (definitions in each case are from left to right).

a) Standard Toolbar:

New Project	Creates a new project.
Open a Project	Opens an existing project (represented by the *project_name*.h3d file)
Save Project	Saves the input data of the current project specified in the main program module if the data were either newly created or changed during an application run.
Project Data Manager	Calls the project manager to manage data of existing projects; helps to locate, open, copy, delete or rename the desired projects and their data (Figs. 2 and 3).
Print (Ctrl+P)	Prints the content of the View window.
Navigator Window	Displays or hides the Navigator Window.
Edit Bar	Displays or hides the Edit Bar.

b) Tools Toolbar:

Undo	Reverses the last edit actions.
Redo	Repeats the last edit actions.
Select by Rhomboid	Selects objects using rhomboid.
Grid and Work Plane Settings	Calls the Grid and Work Plane dialog window (Fig 78).
Show Grid	Shows a Work Plane (axis and origin of the grid) in the View window.
Set Grid Origin	Redefines the origin of the grid.
Snap to Grid	Mouse moves in steps given by the grid.
Set XY-Work Plane	Sets Work Plane to the XY plane.
Set YZ-Work Plane	Sets Work Plane to the YZ plane.
Set XZ-Work Plane	Sets Work Plane to the XZ plane.

c) View Toolbar:

Rotate View	Allows objects in the View windows to be rotated using the mouse while holding the left mouse button.
Scroll View	Allows moving (scrolling) of objects in the View window using the mouse while holding the left mouse button.

Zoom View — Allows zooming of objects in the View window using the mouse while holding the left mouse button.

Similar functions can be achieved by pressing various buttons on the keyboard. **Rotating** is achieved by holding simultaneously the **Ctrl** button on the keyboard and the left mouse button. **Scrolling** occurs by holding simultaneously the **Shift** button on the keyboard and the left mouse button. And finally, **zooming** is achieved by holding simultaneously the **Alt** button on the keyboard and the left mouse button.

Zoom by Rectangle	Zooms in on a certain part of the View window using a rectangle.
View All	Shows the default view of the View window.
Previous view	Shows the previous view of a certain part of the View window.
In Reverse Z-direction	Sets the view of the transport domain in the reverse Z-direction.
View Commands	Shows a pop-up menu with the following commands:

> *Isometric*
> *In X-direction*
> *In Y-direction*
> *In Z-direction*
> *In Reverse X-direction*
> *In Reverse Y-direction*
> *In Reverse Z-direction*
> *Perspective*
> *Default View*

Isometric View	Specifies the isometric view.
Perspective View	Specifies the perspective view.
View Stretching Factors	Calls the View Stretching Factors dialog window (Fig 79).
Rendering Commands	Displays a menu with three commands on how to display the transport domain (see 8.1.4):
Full Model	Displays the transport domain as a full object.
Transparent Model	Displays the transport domain as a transparent object.
Wire Model	Displays the transport domain as a wired object.
Sections Commands	Displays a menu with nine commands for editing sections:
Cut with Rectangle	
Cut with Indexes	
Create New Section from Selection	
Create New Section from Current View	
Display All	
Display Previous	
Display only Selection	
Display Reverse	
Edit Section	
Graph Type Commands	Displays a menu with five commands.
Isolines	Displays the spatial distribution of a certain variable by means of isolines.
Color Contours	Displays the spatial distribution of a certain variable by means of color contours.

Color Points	Displays the spatial distribution of a certain variable by means of color points.
Color Edges	Displays the spatial distribution of a certain variable by means of color edges.
Velocity Vectors	Displays Darcy velocity vectors.
Color Scale Options	Displays menu with seven commands:

Color Smoothing
Surface Lighting
Min/Max Values Global in Time
Min/Max Values Global in Space
Standard Scale
Custom Scale
Edit Scale and Colors

d) GUI Toolbar:

View/Edit Domain Geometry	Sets the View window to View/Edit Domain Geometry mode.
View/Edit FE-Mesh	Sets the View window to View/Edit FE-Mesh mode.
View/Edit Domain Properties	Sets the View window to View/Edit Domain Properties mode, to edit materials.
View/Edit Initial Conditions	Sets the View window to View/Edit Initial Conditions mode, to edit pressure head initial conditions.
View/Edit Boundary Conditions	Sets the View window to View/Edit Boundary Conditions mode, to edit water flow boundary conditions.
Execute Calculation	Executes a HYDRUS version 3.0 FORTRAN application.
View Results	Sets the View window to View Results mode, to view the pressure head distribution.

e) Time Layer Toolbar:
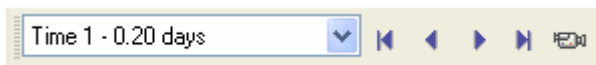
List box of Time Layers	Displays a list box with Time Layers for which results are available.
First Time Layer	Displays the first time layer.
Previous Time Layer	Displays the previous time layer.
Next Time Layer	Displays the next time layer.
Last Time Layer	Displays the last time layer.
Flow Animation	Displays time layers of a particular variable consecutively and continuously.

8.5. *HYDRUS Menus*

The main window of HYDRUS contains a menu that has ten submenus, i.e., File, Edit, View (Fig. 88), Insert, Calculations, Results (Fig. 89), Tools, Options, Windows, and Help (Fig. 90). Table 12 lists the main groups, the main menu items, as well as the main submenu and sub-submenu commands. Table 13 then provides brief descriptions of all menu commands.
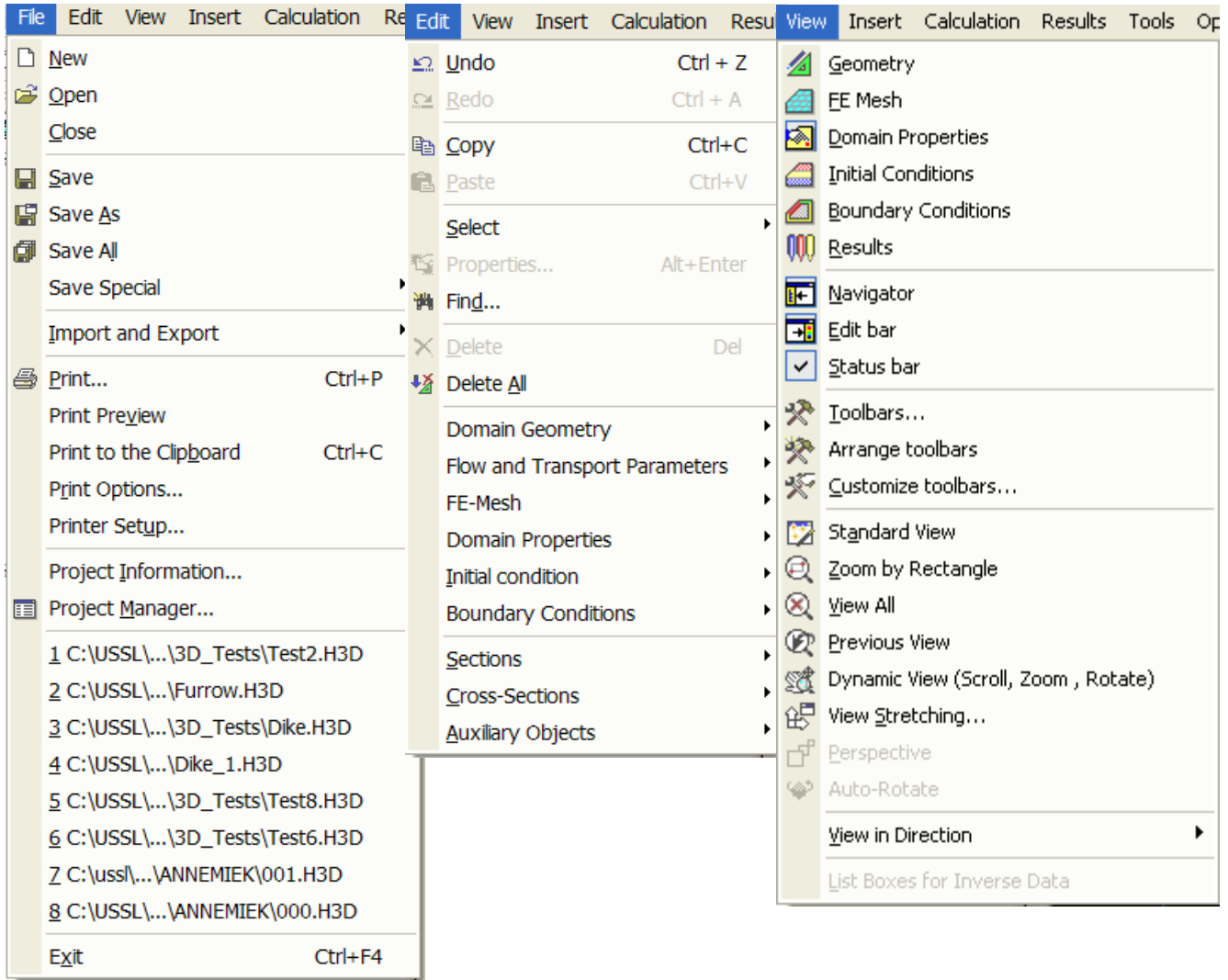
Figure 88. The HYDRUS Menus I (File, Edit, and View).
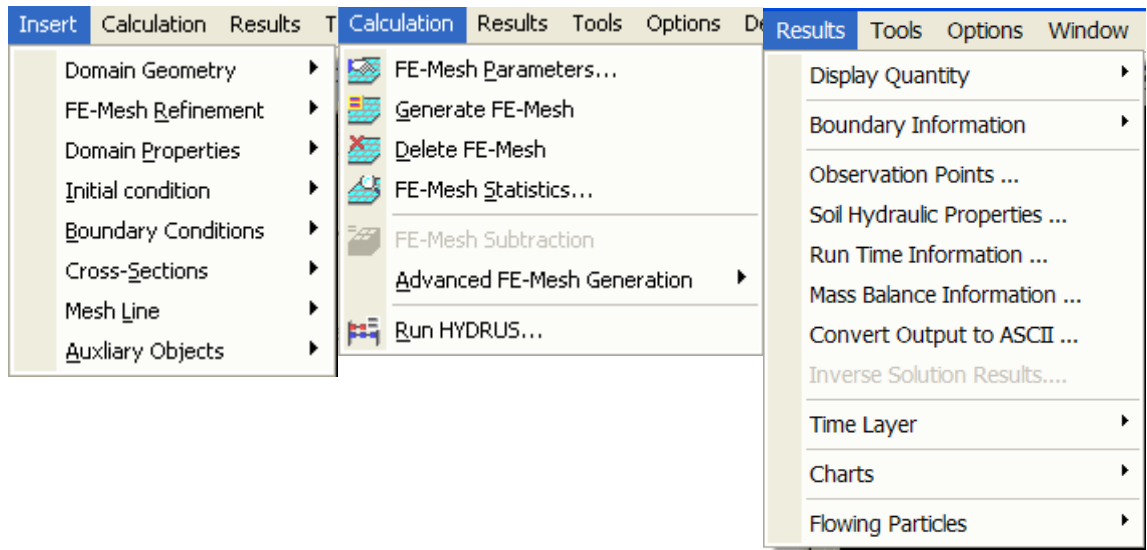
Figure 89. The HYDRUS Menus II (Insert, Calculations, and Results).
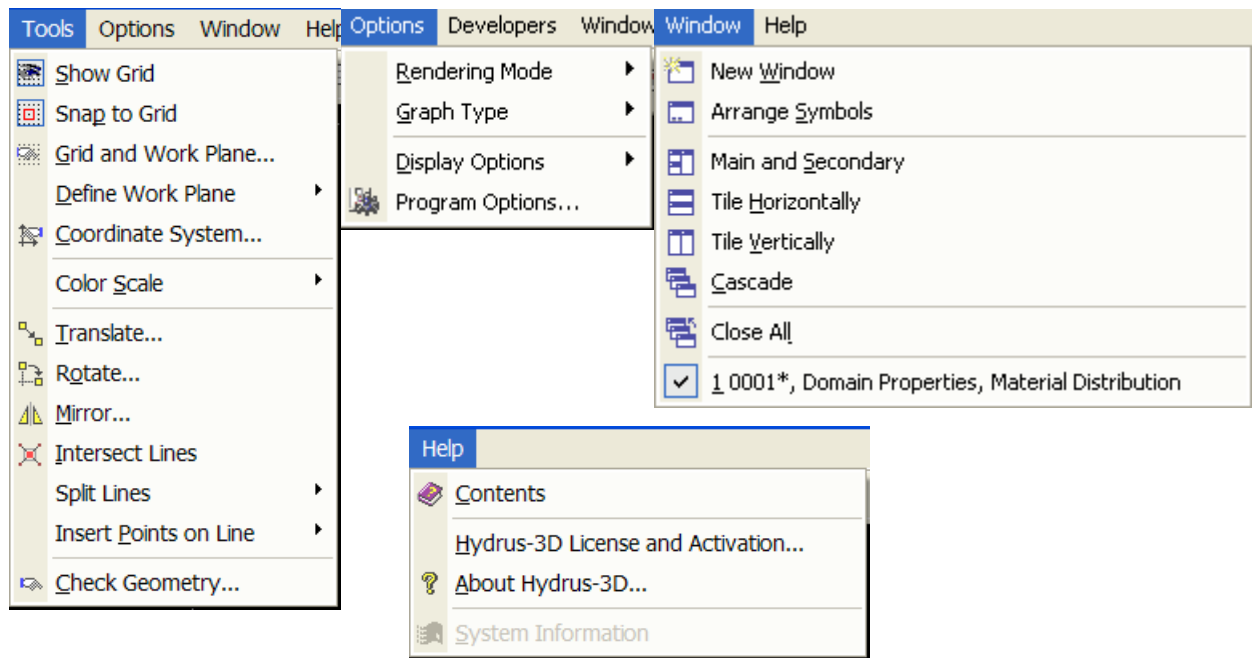
Figure 90. The HYDRUS Menus III (Tools, Options, Windows, and Help).

Table 12. HYDRUS menu commands.

Group	Menu	Submenu	Sub-Submenu
A File	New Open Close		
	Save Save As Save All Save Special	Do not Overwrite TMP files Save to Catalog	
	Import and Export	Import Hydrus-2D Project Import Input Data from *.IN Files Export Data for Hydrus Solver	
	Print Print Preview Print to the Clipboard Print Options Print Setup		
	Project Information Project Manager		
	Recent Files		
	Exit		
B Edit	Undo Redo		
	Copy Paste		
	Select Properties Find	Select by Rhomboid	
	Delete Delete All		
	Domain Geometry	Geometry Information Geometry Definition Points Lines Surfaces Openings Thickness Vectors Solids	
	Flow Parameters	Main Processes Inverse Solution Time Information Output Information Water Flow Parameters	Iteration Criteria Hydraulic Properties Model Soil Hydraulic Parameters

	Solute Transport Parameters	Anisotropy Tensors
		General Information
		Solute Transport Parameters
		Solute Reaction Parameters
		Temperature Dependence
	Heat Transport Parameters	
	Root Water Uptake	Root Water Uptake Models
		Pressure Head Reduction
		Osmotic Head Reduction
	Variable Boundary Condition	
	Data for Inverse Solution	
FE-Mesh	FE-Mesh Generator	
	FE-Mesh Parameters	
	Generate FE-Mesh	
	Delete FE-Mesh	
	FE-Mesh Statistics	
	Advanced FE-Mesh Generation	Fundamental Triangulation
		Mesh Refinement
		Homogeneous Triangulation
		Check of Convexity
		Mesh Smoothing
Domain Properties	Material Distribution	
	Root Distribution	
	Nodal Recharge	
	Scaling Factor	Hydraulic Conductivity
		Pressure Head
		Water Content
	Local Anisotropy	Angle
		First Component
		Second Component
	Subregions	
	Observation Nodes	Edit
		Delete
		Clear All
	Drains	Edit
		Delete
		Clear All
	Flowing Particles	Edit
		Delete
		Clear All
	Stochastic Distribution of S.F.	
	Subregions = Material Distribution	
	Nonequilibrium Conc. = a * Equil. Conc.	
	Parameters for Root Distribution	
	Default Domain Parameters	
Initial Conditions	Pressure Head/Water Content	
	Concentration	
	Nonequilibrium Concentration	
	Temperature	
	Import	
Boundary Conditions	Water Flow	
	Solute Transport	
	Heat Transport	
Sections	Edit Sections	

		New Section from Selection	
		New Section from View	
		Display All	
		Display Previous	
		Display only Selection	
		Display Reverse	
		New Section by Rectangle	
		New Section by Indexes	
	Cross-Sections	Edit	
		Delete Selected	
		Delete All	
	Auxiliary Objects	Dimensions	Delete Selected
			Delete All
		Comments	Edit
			Delete Selected
			Delete All
	Textures		
C View	Geometry		
	FE Mesh		
	Domain Properties		
	Initial Conditions		
	Boundary Conditions		
	Results		
	Navigator		
	Edit Bar		
	Tabs in View		
	Status Bar		
	Toolbars		
	Arrange Toolbars		
	Customize Toolbars		
	Standard View		
	Zoom by Rectangle		
	View All		
	Previous View		
	Dynamic View (Scroll, Zoom, Rotate)		
	View Stretching		
	Perspective		
	Auto-Rotate		
	View in Direction	Isometric	
		In X-direction	
		In Y-direction	
		In Z-direction	
		Reverse X-direction	
		Reverse Y-direction	
		Reverse Z-direction	
	List Boxes for Inverse Data		
D Insert	Domain Geometry	Points	Graphically
			Dialog
			Read from File
		Lines	Line
			Polyline

139

		Arc
		Circle
		Spline
	Surfaces	Graphically
		Graphically - Rectangle
		Dialog
	Openings	Graphically
		Dialog
	Thicknesses	Graphically
		Dialog
	Solids	Graphically
		Graphically - Brick
		Dialog
FE-Mesh Refinement	Graphically	
	Dialog	
Domain Properties	Material Distribution	
	Root Distribution	
	Nodal Recharge	
	Scaling Factor	Hydraulic Conductivity
		Pressure Head
		Water Content
	Local Anisotropy	Angle
		First Component
		Second Component
		Index
	Subregions	
	Observation Nodes	
	Drains	
	Flowing Particles	
Initial Conditions	Pressure Head/Water Content	
	Concentration	
	Nonequilibrium Concentration	
	Temperature	
	Import	
Boundary Conditions	Water Flow	No Flux
		Constant Head
		Constant Flux
		Seepage Face
		Variable Head 1-4
		Variable Flux 1-4
		Free Drainage
		Deep Drainage
		Atmospheric Boundary
	Solute Transport	First-Type
		Third-Type
		Volatile Type
	Heat Transport	First-Type
		Third-Type
Cross-Sections	Graphically	
	Dialog	
Mesh Line	Graphically	
	Dialog	
Auxiliary Objects	Dimension	
	Comment	
	Texture	

E	**Calculation**	FE-Mesh Parameters
		Generate FE-Mesh
		Delete FE-Mesh
		FE-Mesh Statistics
		FE-Mesh Subtraction
		Advance FE-Mesh Generation
		Run HYDRUS

F	**Results**	Display Quantity	Pressure Heads
			Water Content
			Concentration
			Nonequilibrium Concentration
			Temperature
			Velocity
		Boundary Information	Pressure Heads
			Boundary Fluxes
			Cumulative Fluxes
			Solute Fluxes
		Observation Points	
		Soil Hydraulic Properties	
		Run Time Information	
		Mass Balance Information	
		Convert Output to ASCII	
		Inverse Solution Results	
		Time Layer	First
			Last
			Previous
			Next
			Animation
		Charts	Cross-Section
			Boundary Line
			Mesh Line
		Flowing Particles	Draw Particles Positions
			Draw Particles Trajectories

G	**Tools**	Show Grid	
		Snap to Grid	
		Grid and Work Plane	
		Define Work Plane	Set Origin
			Define XY
			Define YZ
			Define XZ
		Coordinate System	
		Color Scale	Standard
			Custom
			Color Smoothing
			Edit Color Scale
		Move/Copy	
		Rotate	
		Mirror	
		Intersect Lines	
		Split Lines	Graphically
			n Pints
			Distance
		Insert Points on Line	Graphically
			n Pints

		Distance
	Check Geometry	
H Options	Rendering Mode	Full Model
		Transparent Model
		Wire Model
	Graph Type	Izolines
		Color Contours
		Color Points
		Color Edges
		Velocity Vectors
	Display Options	Edit
		Default
		Read
		Save As
	Program Options	
I Windows	New Window	
	Arrange Symbols	
	Main and Secondary	
	Tile Horizontally	
	Tile Vertically	
	Cascade	
	Close All	
J Help	Contents	
	Hydrus License and Activation	
	About Hydrus	
	System Information	

Table 13. Brief description of HYDRUS menu commands.

Group	Command	Brief description of the command
A	**File**	
	New	Creates a new project after user provides required information (name and description) in the **New Project** dialog window (Fig. 4). Project will be located in the current Workspace.
	Open	Opens an existing project (represented by the *project_name*.h3d file) using the **Open** dialog window, which a user uses to browse for the HYDRUS project.
	Close	Closes an open project.
	Save	Saves the input data of an actual project specified in the main program module if the data were either newly created or changed during an application run.
	Save As	Saves data of a particular project under a new project name using **Save As** dialog window.
	Save All	Saves data of all projects.
	Save Special	
	Do not Overwrite TMP files	Saves the project without overwriting temporary files.
	Save to Catalog	Saves the project to catalog from which projects can be used in other applications.
	Import and Export	
	Import Hydrus-2D Project	Imports Hydrus-2D projects from Version 2.0.
	Import Input Data from *.IN Files	Import input data from *.in files that may have been modified manually outside of the HYDRUS GUI.
	Export Data for Hydrus Solver	Export data for the HYDRUS solver into the working directory from which the computational module reads inputs and into which it writes outputs.
	Print	Prints the content of the View window.
	Print Preview	Previews the content of the View window before printing.
	Print to the Clipboard	Copies the content of the View window to the clipboard for subsequent pasting into other software packages.
	Print Options	Select various print options, such as print quality, text size, frame, and text content (Fig. 94).
	Print Setup	Selects a printer and printer connection.
	Project Information	Displays information about the current project (in the **General Data** dialog window (Fig. 4)).
	Project Manager	Calls the **Project Manager** (Figs. 2 and 3) to manage data of existing projects; helps to locate, open, copy, delete or rename the desired projects and their data.
	Recent Files	Displays recently opened projects.
	Exit	Closes open projects and leaves the program. This command informs users before exiting the application whether or not the input data of open projects were changed during the application run. If changes did occur, users are given an option to save data before exiting the application.
B	**Edit**	

Undo	Reverses the last edit action.
Redo	Repeats the last edit action.
Copy	Copies a selected object.
Paste	Pastes a selected object.
Select by Rhomboid	Selects objects by means of a rhomboid.
Properties	Displays properties of a selected object.
Find	Finds an object.
Delete	Deletes a selected object.
Delete All	Deletes all objects.

Domain Geometry

Geometry Information	Specifies whether the flow and transport problem occurs in a two- or three-dimensional transport domain, and whether the domain is simple or complex using the **Geometry Information** dialog window (Fig. 6).
Geometry Definition	Specifies parameters (dimensions and slopes) for simple rectangular or hexahedral transport domains using the **Rectangular** (Fig. 10) or **Hexahedral Domain Definition** (Fig. 11) dialog windows.
Points	Edits selected points.
Line	Edits selected lines using the **Edit Curve** dialog window (Fig. 35).
Surfaces	Edits selected surfaces.
Openings	Edits selected openings.
Thickness Vectors	Edits selected thickness vectors.
Solids	Edits solids.

Flow Parameters

Main Processes	Selects the processes to be simulated, i.e., water flow, multiple solute transport, heat transport, and/or root water uptake (the **Main Processes** dialog window (Fig. 12)).
Inverse Solution	Selects type of weighting of measured data, and whether soil hydraulic parameters, solute transport parameters, and/or heat transport parameters, are to be fitted (the **Inverse Solution** dialog window (Fig. 13)).
Time Information	Selects time units, and gives the time discretization information (the **Time Information** dialog window (Fig. 15)).
Output Information	Specifies print options (The **Output Information** dialog window (Fig. 16)).

Water Flow Parameters

Iteration Criteria	Specifies iteration criteria for the solution precision and parameters for the time step control (The **Iteration Criteria** dialog window (Fig. 17)).
Hydraulic Properties Model	Selects the type of model used for the soil hydraulic properties, and decides whether hysteresis is to be considered (the **Soil Hydraulic Model** dialog window (Fig. 18)).
Soil Hydraulic Parameters	Specifies parameters in the soil hydraulic model (the **Water Flow Parameters** dialog window (Fig. 19)).
Anisotropy Tensors	Defines anisotropy tensors for three-dimensional applications (the **Tensors of Anisotropy** dialog window (Fig. 21)).

Solute Transport Parameters

General Information	Selects the time and spatial weighting schemes for numerical solution of the solute transport equation; specifies the number of solutes to be considered (the **Solute Transport** dialog window (Fig. 22)).

Solute Transport Parameters	Specifies solute transport parameters (the **Solute Transport Parameters** dialog window (Fig. 23)).
Solute Reaction Parameters	Specifies solute reaction parameters (the **Solute Reaction Parameters** dialog window (Fig. 24)).
Temperature Dependence	Specifies parameters defining the temperature dependence of reaction and transport parameters (the Temperature Dependent Solute Transport and Reaction Parameters dialog window shown in Figure 25).
Heat Transport Parameters	Specifies heat transport parameters (the **Heat Transport** dialog window; Fig. 26).

Root Water Uptake

Root Water Uptake Models	Selects the root water uptake stress response models for both salinity and water stress (the **Root Water Uptake Model** dialog window, Fig. 27).
Pressure Head Reduction	Specifies parameters in the root water uptake water stress response model (the **Root Water Uptake Parameters** dialog window; Fig. 28).
Osmotic Head Reduction	Specifies parameters in the root water uptake salinity stress response model (the **Root Water Uptake Parameters** dialog window; Fig. 29).
Variable Boundary Condition	Specifies time-dependent boundary conditions for all transport processes (the **Time Variable Boundary Conditions** dialog window; Fig. 31).
Data for Inverse Solution	Specifies data for the inverse solution, their type, location and associated weight (the **Data for Inverse Solution** dialog window; Fig. 14).

FE-Mesh

FE-Mesh Generator	Selects the structured or unstructured finite element mesh generator (the **Finite Element Mesh Generator** dialog window; Fig. 56).
FE-Mesh Parameters	Specifies either parameters of the Unstructured Finite Element Mesh Generator (the **FE-Mesh Parameters** dialog window; Figs. 59 through 62) or parameters of the structured mesh (the **Rectangular Domain Discretization** dialog window (Fig. 57) or the **Hexahedral Domain Discretization** dialog window (Fig. 58)).
Generate FE-Mesh	Generates unstructured finite element mesh.
Delete FE-Mesh	Deletes unstructured finite element mesh.
FE-Mesh Statistics	Provides information about finite element mesh (the **FE-Mesh Information** dialog window (Fig. 66).

Advanced FE-Mesh Generation

Fundamental Triangulation	Performs triangulation of boundary nodes based on the Delaunay criterion.
Mesh Refinement	Inserts a new point in the center of all triangles that do not fulfill the smoothness criterion.
Retriangulation	Retriangulates mesh according to Delaunay criterion.
Check of Convexity	Corrects possible errors which may appear during smoothing and retriangulating.
Mesh Smoothing	Smoothes the mesh by solving a set of coupled elliptic equations using a recursive algorithm.

Domain Properties

Material Distribution	Specifies the spatial distribution of soil materials.
Root Distribution	Specifies the spatial distribution of root water uptake.
Nodal Recharge	Specifies the spatial distribution of nodal recharge.

Scaling Factor

Hydraulic Conductivity	Specifies the spatial distribution of hydraulic conductivity scaling factors.
Pressure Head	Specifies the spatial distribution of pressure head scaling factors.
Water Content	Specifies the spatial distribution of water content scaling factors.

Local Anisotropy

Angle	Specifies the spatial distribution of the angle of local anisotropy.
First Component	Specifies the spatial distribution of the first component of local anisotropy.
Second Component	Specifies the spatial distribution of the second component of local anisotropy.
Index	Specifies the spatial distribution of the index that represents the local anisotropy tensor.
Subregions	Specifies the spatial distribution of subregions for the mass balance calculations.

Observation Nodes

Edit	Specifies observation nodes for output of the pressure head, water content, temperature, and concentration at each time step.
Delete	Deletes selected observation nodes.
Clear All	Deletes all observation nodes.

Drains

Edit	Specifies nodal points representing tile drains.
Delete	Deletes selected tile drains.
Clear All	Deletes all tile drains.

Flowing Particles

Edit	Specifies nodal points representing flowing particles.
Delete	Deletes selected flowing particles.
Clear All	Deletes all flowing particles.

Stochastic Distribution of S.F.	Generates a stochastic distribution of scaling factors.
Subregions = Material Distribution	Makes the subregions for mass balance calculations similar to those for the soil materials.
Nonequilibrium Conc. = a * Equil. Conc.	Specifies that nonequilibrium concentrations (i.e., kinetically sorbed concentrations or concentrations in the immobile water) are a multiple of the liquid phase concentrations.
Parameters for Root Distribution	Specifies parameters for the spatial distribution of root water uptake.
Default Domain Parameters	Specifies default domain properties that are constant for the same depths (the **Default Domain Properties** dialog window; Fig. 68).

Initial Conditions

Pressure Head/Water Content	Specifies the initial condition for water flow.
Concentration	Specifies the initial condition for solute transport (equilibrium concentrations).
Nonequilibrium Concentration	Specifies the initial condition for solute transport (nonequilibrium concentrations).
Temperature	Specifies the initial condition for heat transport.
Import	Imports the initial condition from previous simulations for water flow, and solute and heat transport.

Boundary Conditions

Water Flow	Specifies boundary conditions for water flow.
Solute Transport	Specifies boundary conditions for solute transport.
Heat Transport	Specifies boundary conditions for heat transport.

Sections

Edit Sections	Calls the FE-Mesh Section dialog (Fig. 67). HYDRUS recognizes two different definitions of Sections – one for geometrical objects and one for the FE-mesh. A different dialog appears when called from the "Domain Geometry" part of the program. In all other cases, the FE-mesh Section dialog appears.
New Section from Selection	Creates a new section from currently selected objects/FE-Mesh
New Section from View	Creates a new section from currently displayed objects/FE-Mesh
Display All	Displays all objects or entire FE-Mesh
Display Previous	Displays a previously displayed view (sections)
Display only Selection	Displays only currently selected objects/FE-Mesh
Display Reverse	Hides currently displayed objects/FE-Mesh and displays currently hidden objects/FE-Mesh
Cut with &Rectangle	Displays objects/FE-mesh nodes within a certain rectangle or rhomboid and hides all the others
Cut with &Indexes	Displays objects with given indexes and hides all the others

Cross-Sections

Edit	Edits a cross-section
Delete Selected	Deletes selected cross-sections
Delete All	Deletes all cross-sections

Auxiliary Objects
Dimensions

Delete Selected	Deletes selected dimensions
Delete All	Deletes all dimensions

Comments

Edit	Edits a comment
Delete Selected	Deletes selected comments
Delete All	Deletes all comments

C   View

Geometry	Sets the View window to View/Edit Domain Geometry mode.
FE Mesh	Sets the View window to View/Edit FE-Mesh mode.
Domain Properties	Sets the View window to View/Edit Domain Properties mode, to edit materials.
Initial Conditions	Sets the View window to View/Edit Initial Conditions mode, to edit pressure head initial conditions.
Boundary Conditions	Sets the View window to View/Edit Boundary Conditions mode, to edit water flow boundary conditions.
Results	Sets the View window to View Results mode, to view pressure head distribution.
Navigator	Displays or hides the Navigator window.
Edit Bar	Displays or hides the Edit Bar.
Tabs in View	Displays or hides Tabs in the View window.
Status Bar	Displays or hides the Status Bar.
Toolbars	Selects which toolbars are to be displayed (the **Toolbars** dialog window, Fig. 86).
Arrange Toolbars	Arranges toolbars.
Customize Toolbars	Customizes toolbars (the **Customize Toolbars** dialog window, Fig. Fig. 87).
Standard View	Sets a default viewing direction (in 3D) and performs the "View All" command
Zoom by Rectangle	Zooms in on a certain part of the View window using a rectangle.

View All	Changes a scroll position and a zoom factor so that all currently displayed objects are visible in the View Window. This command does not change the viewing direction.
Previous View	Shows the previous view on a certain part of the View window.
Dynamic View (Scroll, Zoom, Rotate)	Sets the View windows so that dynamic actions can be carried out with a cursor. Moving the cursor while holding the left mouse button allows the object to be displayed in a different part of the View window. Pressing the Shift button on the keyboard allow zooming actions around the cursor.
View Stretching	Calls the View Stretching Factors dialog window (Fig. 79) and adjust stretching factors.
Perspective	Sets perspective view.
Auto-Rotate	Starts *Autorotate* function that will rotate the transport domain in the View window.

View in Direction

Isometric	Sets isometric view.
In X-direction	Sets the view of the transport domain in the *X*-direction.
In Y-direction	Sets the view of the transport domain in the *Y*-direction.
In Z-direction	Sets the view of the transport domain in the *Z*-direction.
Reverse X-direction	Sets the view of the transport domain in the reverse *X*-direction.
Reverse Y-direction	Sets the view of the transport domain in the reverse *Y*-direction.
Reverse Z-direction	Sets the view of the transport domain in the reverse *Z*-direction.
List Boxes for Inverse Data	Shows text information in the inverse data list (the **Data for Inverse Solution** dialog window; Fig. 14).

D   Insert

Domain Geometry
Points

Graphically	Inserts single points graphically.
Dialog	Inserts single points numerically (the **Edit Point** dialog window without the FE-Mesh tab; Fig. 34).
Read From File	Reads single point coordinates from a file.

Lines

Line	Inserts a line either graphically or numerically (the **Edit Curve** dialog window without the FE-Mesh tab; Fig. 35).
Polyline	Inserts a polyline either graphically or numerically (the **Edit Curve** dialog window without the FE-Mesh tab; Fig. 35).
Arc	Inserts an arc either graphically or numerically.
Circle	Inserts a circle either graphically or numerically.
Spline	Inserts a spline either graphically or numerically (the **Edit Curve** dialog window without the FE-Mesh tab; Fig. 35).
Surfaces	Inserts a surface either graphically or numerically.
Openings	Inserts an opening either graphically or numerically.
Thicknesses	Inserts thicknesses either graphically or numerically.
Solids	Inserts solids either graphically or numerically.

FE-Mesh Refinement

Graphically	Inserts new FE-mesh refinement graphically (the **FE-Mesh refinement** dialog window; Fig. 63).
Dialog	Defines new FE-mesh refinement graphically (the **FE-Mesh refinement** dialog window; Fig. 63).

Domain Properties

Material Distribution	Specifies the spatial distribution of soil materials.

Root Distribution	Specifies the spatial distribution of root water uptake.
Nodal Recharge	Specifies the spatial distribution of nodal recharge.

Scaling Factor

Hydraulic Conductivity	Specifies the spatial distribution of the hydraulic conductivity scaling factors.
Pressure Head	Specifies the spatial distribution of the pressure head scaling factors.
Water Content	Specifies the spatial distribution of the water content scaling factors.

Local Anisotropy

Angle	Specifies the spatial distribution of the angle of local anisotropy (for two-dimensional applications).
First Component	Specifies the spatial distribution of the first component of local anisotropy (for two-dimensional applications).
Second Component	Specifies the spatial distribution of the second component of local anisotropy (for two-dimensional applications).
Index	Specifies the spatial distribution of anisotropy tensors (for three-dimensional applications).
Subregions	Specifies the spatial distribution of subregions for mass balance calculations.
Observation Nodes	Specifies observation nodes for output of the pressure head, water content, temperature, and concentration at each time step.
Drains	Specifies nodal points representing tile drains.
Flowing Particles	Specifies nodal points representing flowing particles.

Initial Conditions

Pressure Head/Water Content	Specifies the initial condition for water flow.
Concentration	Specifies the initial condition for solute transport.
Nonequilibrium Concentration	Specifies the initial condition for nonequilibrium solute transport.
Temperature	Specifies the initial condition for heat transport.
Import	Imports initial conditions for water flow, solute transport, and/or heat transport.

Boundary Conditions
Water Flow

No Flux	Specifies a no-flux boundary condition along a selected part of the boundary.
Constant Head	Specifies a constant pressure head boundary condition along a selected part of the boundary.
Constant Flux	Specifies a constant flux boundary condition along a selected part of the boundary.
Seepage Face	Specifies a seepage face boundary condition along a selected part of the boundary.
Variable Head 1-4	Specifies a variable pressure head boundary condition along a selected part of the boundary.
Variable Flux 1-4	Specifies a variable flux boundary condition along a selected part of the boundary.
Free Drainage	Specifies a free drainage boundary condition along a selected part of the boundary.
Deep Drainage	Specifies a deep drainage boundary condition along a selected part of the boundary.
Atmospheric Boundary	Specifies an atmospheric boundary condition along a selected part of the boundary.

Solute Transport

First-Type	Specifies a first-type boundary condition for solute transport along a selected part of the boundary.

Third-Type		Specifies a third-type boundary condition for solute transport along a selected part of the boundary.
Volatile Type		Specifies a volatile-type boundary condition for solute transport along a selected part of the boundary.

Heat Transport

First-Type		Specifies a first-type boundary condition for heat transport along a selected part of the boundary.
Third-Type		Specifies a third-type boundary condition for heat transport along a selected part of the boundary.

Cross-Sections

Graphically	Inserts a cross-section graphically.
Dialog	Inserts a cross-section using the dialog window.

Mesh Line

Graphically	Inserts a mesh line graphically.
Dialog	Inserts a mesh line using the dialog window.

Auxiliary Objects

Dimension	Inserts auxiliary object (dimensions).
Comment	Inserts auxiliary object (dimensions).
Texture	Inserts auxiliary object (texture).

E   Calculation

FE-Mesh Parameters	Specifies either parameters of the Unstructured Finite Element Mesh Generator (the **FE-Mesh Parameters** dialog window; Figs. 59 through 62) or parameters of the structured mesh (the **Rectangular Domain Discretization** dialog window (Fig. 57) or the **Hexahedral Domain Discretization** dialog window (Fig. 58)).
Generate FE-Mesh	Generates the unstructured finite element mesh.
Delete FE-Mesh	Deletes the unstructured finite element mesh.
FE-Mesh Statistics	Provides information about the finite element mesh (the **FE-Mesh Information** dialog window (Fig. 66).

Advanced FE-Mesh Generation

Fundamental Triangulation	Performs triangulation of boundary nodes based on the Delaunay criterion.
Mesh Refinement	Inserts a new point in the center of all triangles that do not fulfill the smoothness criterion.
Retriangulation	Retriangulates mesh according to Delaunay criterion.
Check of Convexity	Corrects possible errors which may appear during smoothing and retriangulating.
Mesh Smoothing	Smoothes the mesh by solving a set of coupled elliptic equations using a recursive algorithm.
Run HYDRUS	Executes a HYDRUS version 3.0 FORTRAN application.

F   Results

Display Quantity

Pressure Head	Displays results in terms of pressure heads.
Water Content	Displays results in terms of water contents.
Velocity	Displays results in terms of velocities.
Concentration	Displays results in terms of concentrations
Nonequilibrium Concentration	Displays results in terms of nonequilibrium concentrations (kinetically sorbed or in the immobile water)
Temperature	Displays results in terms of temperatures

Boundary Information

Pressure Heads	Graphical presentation of pressure heads at different boundaries and in the root zone.
Boundary Fluxes	Graphical presentation of potential and actual boundary water fluxes at different boundaries.
Cumulative Fluxes	Graphical presentation of potential and actual cumulative boundary water fluxes.
Solute Fluxes	Graphical presentation of actual and cumulative boundary solute fluxes.
Observation Points	Graphical presentation of changes in water content, pressure head, temperature, and/or solute and sorbed concentration at specified observation nodes.
Soil Hydraulic Properties	Graphical presentation of the soil hydraulic properties.
Run Time Information	Graphical presentation of information about the number of iterations, time step, and Peclet and Courant numbers.
Mass Balance Information	Displays mass balance information and mean profile properties.
Convert Output to ASCII	Convert binary input and output files into ASCII files.
Inverse Solution Results	Displays information about the inverse solution.

Time Layer

First	Displays a particular variable at the first time layer.
Last	Displays a particular variable at the last time layer.
Previous	Displays a particular variable at the previous time layer.
Next	Displays a particular variable at the next time layer.
Animation	Displays time layers of a particular variable consecutively and continuously.

Charts

Cross-Section	Displays values of a particular variable along an arbitrary cross-section.
Boundary Line	Displays values of a particular variable along a certain part of a boundary.
Mesh Line	Displays values of a particular variable along a selected mesh line.

Flowing Particles

Draw Particles Positions	Draws positions of flowing particles.
Draw Particles Trajectories	Draws trajectories of flowing particles.

G   Tools

Show Grid	Shows or hides the grid.
Snap to Grid	Specifies whether or not the mouse should move in steps defined by the grid.
Grid and Work Plane	Calls the **Grid and Work Plane** dialog window (Fig 78).

Define Work Plane

Set Origin	Redefines origin of the grid.
Define XY	Sets Work Plane to the XY plane.
Define YZ	Sets Work Plane to the YZ plane.
Define XZ	Sets Work Plane to the XZ plane.
Coordinate System	Selects coordinate system

Color Scale

Standard	Selects a standard color scale for the display of a particular variable.
Custom	Selects a custom color scale for the display of a particular variable.
Color Smoothing	Changes color from abrupt to gradual at isolines.
Edit Color Scale	Calls the **Edit Isoband Value and Color Spectra** dialog window (Fig 75).

Move/Copy	Moves or copies a selected object.
Rotate	Rotates a selected object.
Mirror	Mirrors a selected object
Intersect Lines	Finds the intercept of two lines and insert an interception point on the lines.
Split Lines	Splits lines.
Insert Points on Line	Inserts points on a line.
Check Geometry	Checks geometry for consistency.

H   Options

Rendering Mode

Full Model	Displays the transport domain as a full object.
Transparent Model	Displays the transport domain as a transparent object.
Wire Model	Displays the transport domain as a wired object.

Graph Type

Isolines	Displays the spatial distribution of a particular variable by means of isolines.
Color Contours	Displays the spatial distribution of a particular variable by means of isobands.
Color Points	Displays the spatial distribution of a particular variable by means of color points.
Color Edges	Displays the spatial distribution of a particular variable by means of color edges.
Velocity Vectors	Displays Darcy velocity vectors.

Display Options

Edit	Edits display options in the **Display Options** dialog window (Fig 74).
Default	Sets display options to their default values.
Read	Reads display options from a file.
Save As	Saves display options to a file.
Program Options	Displays program options information (the **Program Options** dialog window has two tabs, one related to **Graphics** (Fig. 91) and one to **Program** itself (Fig. 92)).

I   Windows

New Window	Open a new View window.
Arrange Symbols	Arranges minimized windows as icons at the bottom of the **View** window.
Main and Secondary	Displays open View windows as main and secondary windows.
Tile Horizontally	Tiles open View windows horizontally.
Tile Vertically	Tiles open View windows vertically.
Cascade	Cascades open View windows.
Close All	Closes all open View windows.

J   Help

Contents	Displays help information.
Hydrus License and Activation	Displays the Hydrus license and activation information (the **HYDRUS License and Activation** dialog window; Fig. 93).
About Hydrus	Displays the version and authors of the HYDRUS application.
System Information	Displays system information.

9. Miscellaneous Information

9.1. *Program Options*

The **Program Options** dialog window has two tabs, one related to **Graphics** (Fig. 91) and one to **Program Directories** (Fig. 92).

On the left side of the **Graphics** Tab (*OpenGL*) one can turn on or off the *OpenGL Hardware Acceleration* (OpenGL is a library of functions developed by Silicon Graphics Inc. for handling graphical objects), and select the speed for *OpenGL optimization*.

On the right side of the **Graphics** Tab (*Options*) one can select
a) *Simplified display in Move modus*
b) whether or not an object is selected when the cursor hovers above it (*Pre-selection (Mark object while hovering above it with cursor)*), and
c) whether or not values and properties are displayed numerically when the cursor is close to a selected object (*Display values/properties at pre-selected objects*),
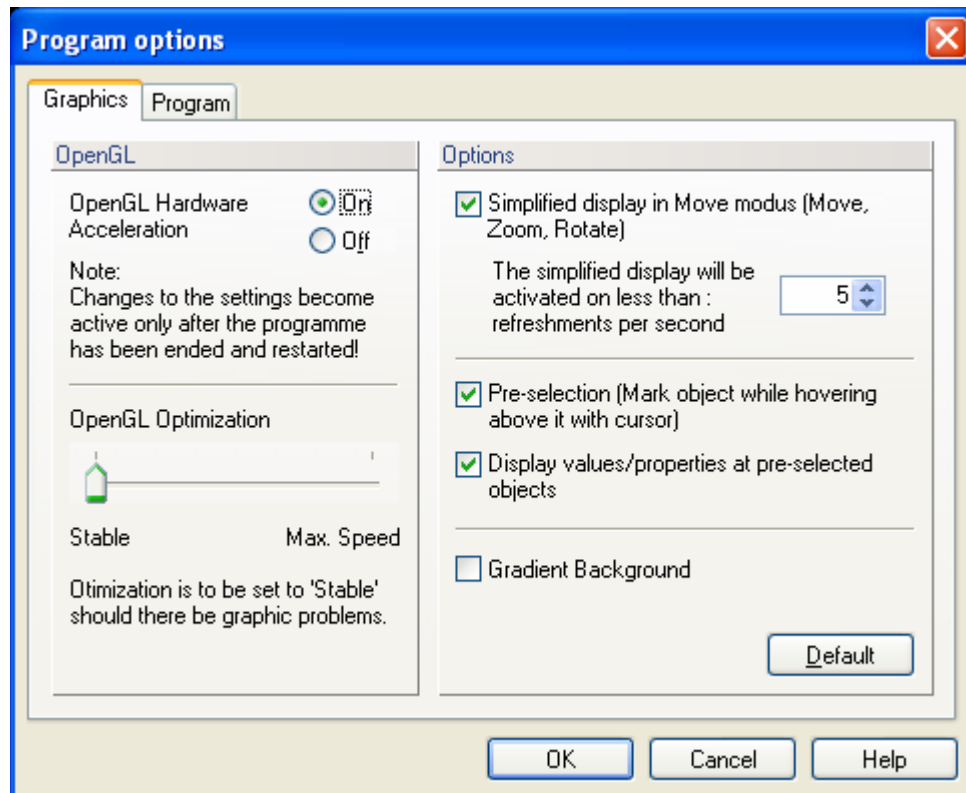d) a different background (*Gradient Background*),

Figure 91. The Program Options dialog window (the Graphics Tab).

On the left side of the **Program Directories** Tab (*Options*) one can
   a) select a time interval for *Auto-save*
   b) specify *Memory size for the Undo buffer*, and
   c) specify the *Maximum number of FE-Mesh*.

On the right side of the **Program Directories** Tab (*Options*) one can
   a) specify whether or not the program *Reloads last opened projects at startup*
   b) select whether the FE-Mesh is to be saved in text format (*Save FE-Mesh in text format*)
   c) select whether or not Domain Properties are to be saved in text format (*Save Domain Properties in text format*), and
   d) specify whether the results are to be kept in an external directory (*By default keep results in external directory*).

One can also specify in this dialog window the *Working directory for temporary files* and the *Configuration file for display options*.
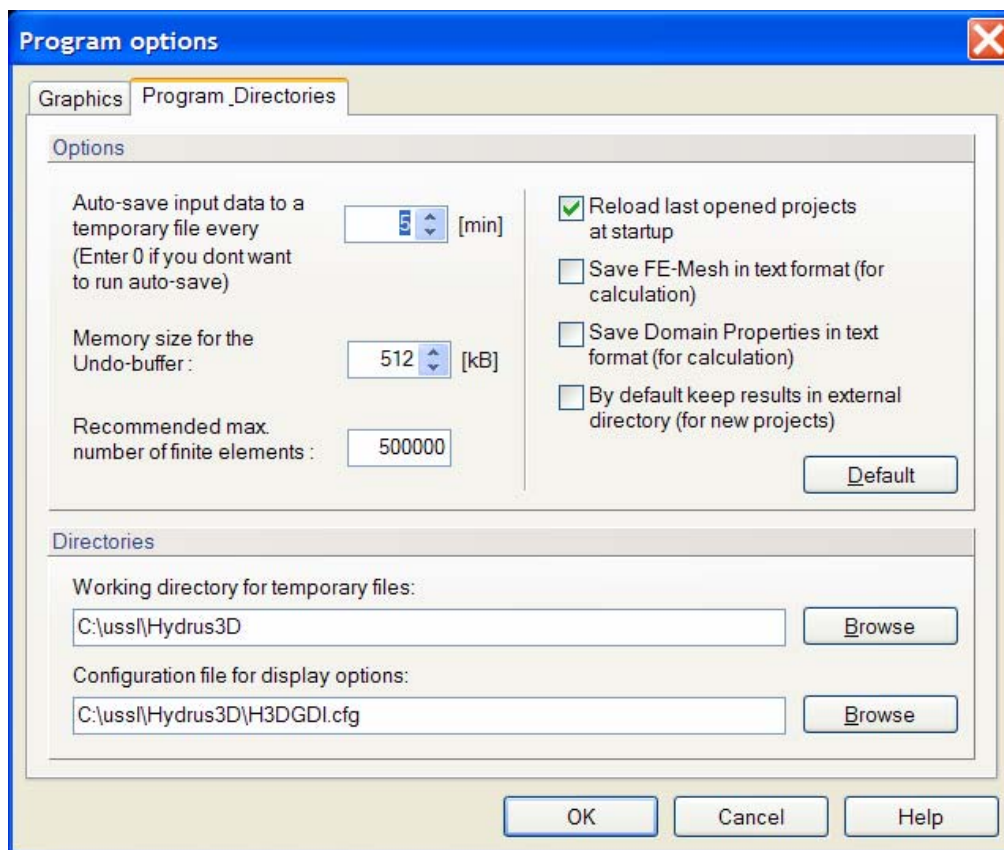

Figure 92. The Program Options dialog window (the Program Directories Tab).

9.2. *HYDRUS License and Activation*

The HYDRUS software package is activated using the **HYDRUS License and Activation** dialog window (Fig. 93) that is called using the command *Help->Hydrus License and Activation*.

Figure 93. The HYDRUS License and Activation dialog window.

9.3. *Print Options Dialog Window*

The **Print Options** dialog window contains three tabs (Fig. 94). In the **General Tab** a user selects whether the content of the View window (*Picture*) is to be printed with or without a **Legend**, **Page Orientation** (Portrait or Landscape), and **Page Margins**. In the **Picture Tab**, users further select **Print Quality**, whether the **Frame** is to be printed in black or color (**Colors and Frame**), and **Text Size**. Finally, in the **Legend Tab**, users select what texts **(Legend Rows)** are to be printed, with what **Font**, and how far from the picture. Users can use a predefined text or can write their own.
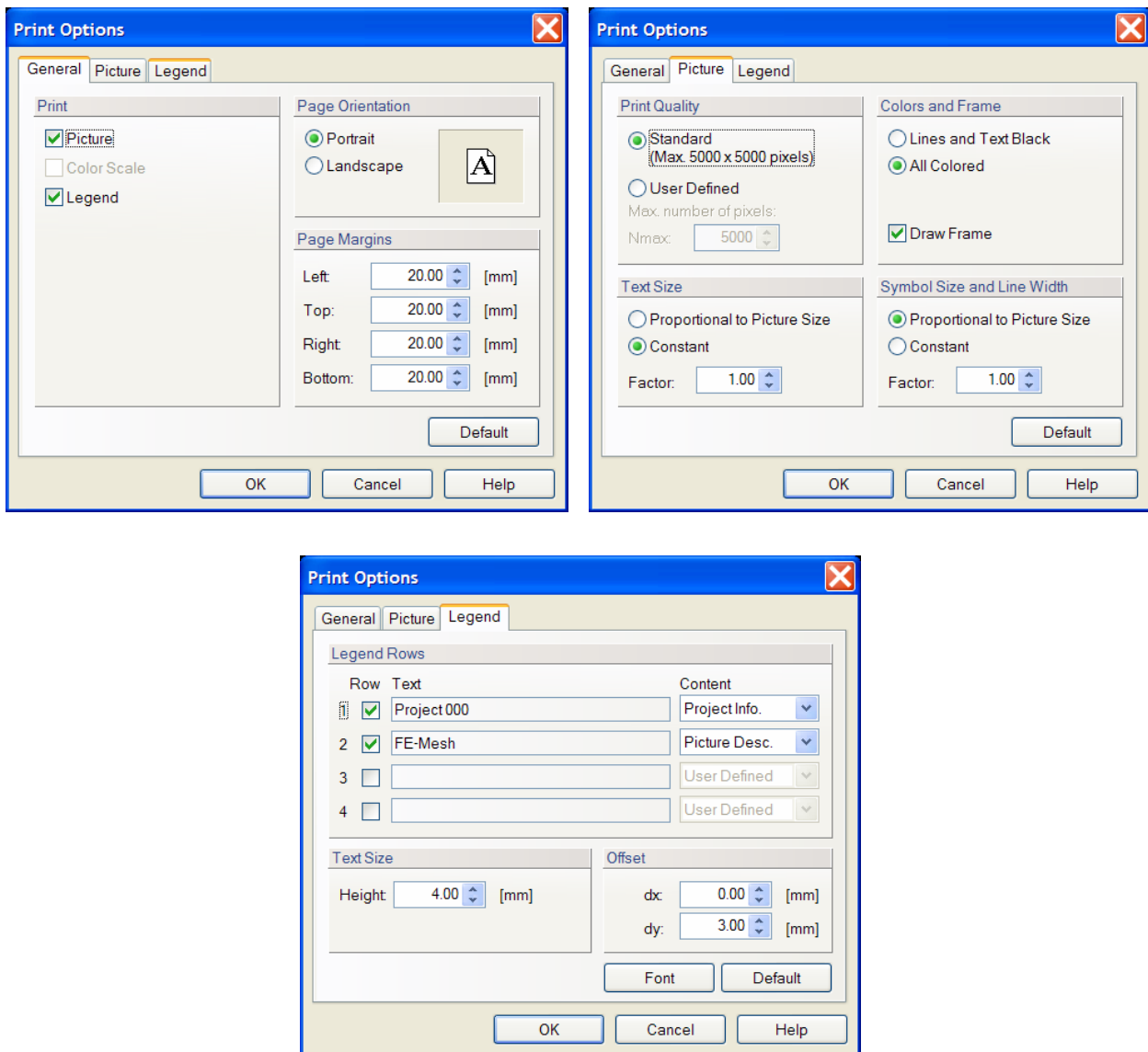

Figure 94. The General, Picture, and Legend tabs of the Print Options dialog window.

156

9.4. *Coordinate systems*

The **Coordinate System** to be used for the transport domain definition can be selected from the **List of Available Coordinate Systems** dialog window (Fig. 95, top) called by the *Tools->Coordinate System* command. **Cartesian**, **Cylindrical**, and **Spherical** systems are available. The **Cartesian** coordinate system is selected by default. A **New Coordinate System** can be defined using the dialog window of the same name (Fig. 95, bottom).
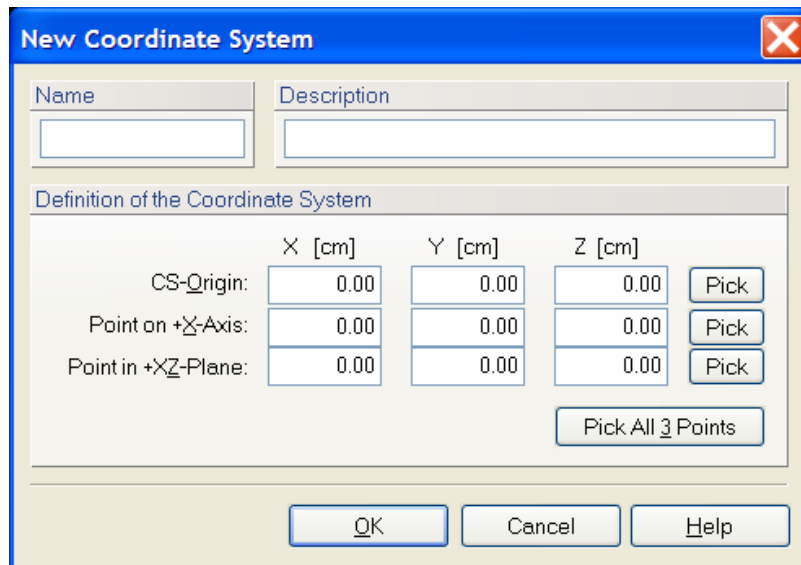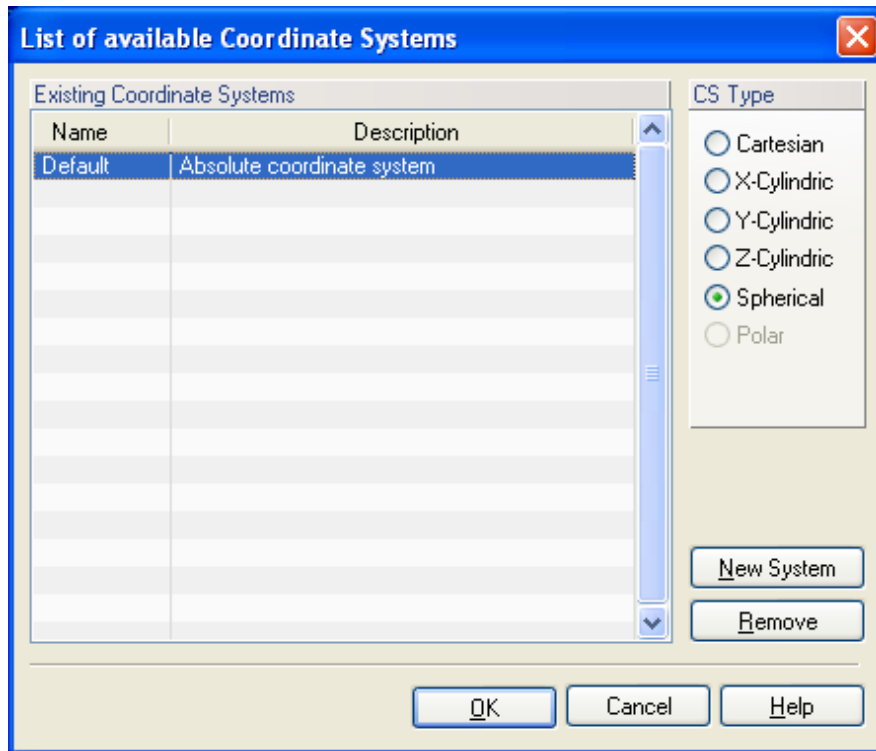
Figure 95. The Coordinate Systems dialog windows.

9.5. *DOS Window During Calculations*

During the calculations different type of information can be written to the screen. The following information may be written to the screen depending upon the problem:

Time	Time
T-Level	Time level
ItW	Number of iterations to solve the water flow problem at a certain time step
ItC	Number of iterations to solve the solute transport problem at a certain time step
ItCum	Cumulative number of iterations
CumAtmBC	Cumulative flux across the atmospheric boundary
CumConst	Cumulative flux across the boundary having a constant flux or pressure head
CumDrain	Cumulative flux across the boundary having a time-variable flux or pressure head
CumRootUp	Cumulative actual root water uptake
CumCh0	Cumulative zero-order production in domain (solute transport)
CumCh1	Cumulative first-order degradation in domain (solute transport)
CumChS	Cumulative solute flux across the boundary having a time-variable flux or pressure head
hAtm	The average pressure head at the atmospheric boundary
hConst	The average pressure head at the boundary having a constant flux or pressure head
hDrain	The average pressure head at the boundary having a time-variable flux or pressure head
hRoot	The average pressure head in the root zone
hSeep	The average pressure head at the boundary with the seepage face
vConstBC	Flux across the boundary having a constant flux or pressure head
vSeep	Flux across the boundary having a seepage face

References

Brooks, R. H., and A. T. Corey, Properties of porous media affecting fluid flow. *J. Irrig. Drainage Div.*, ASCE Proc. 72(IR2), 61-88, 1966.

Carsel, R.F., and Parrish, R. S., Developing joint probability distributions of soil water retention characteristics, *Water Resour. Res*, *24*, 755-769, 1988.

Chung S.-O., and R. Horton, Soil heat and water flow with a partial surface mulch, *Water Resour. Res., 23*(12), 2175-2186, 1987.

Durner, W., Hydraulic conductivity estimation for soils with heterogeneous pore structure, *Water Resour. Res*., *32*(9), 211-223, 1994.

Feddes, R. A., P. J. Kowalik, and H. Zaradny, *Simulation of Field Water Use and Crop Yield*, John Wiley & Sons, New York, NY, 1978.

Hopmans, J. W., J. Šimůnek, N. Romano, and W. Durner, Inverse Modeling of Transient Water Flow, In: *Methods of Soil Analysis, Part 1, Physical Methods,* Chapter 3.6.2, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 963-1008, 2002.

Kool, J. B., and J. C. Parker, Development and evaluation of closed-form expressions for hysteretic soil hydraulic properties. *Water Resour. Res., 23*(1), 105-114, 1987.

Kosugi K., Lognormal distribution model for unsaturated soil hydraulic properties, *Water Resour. Res*., *32*(9), 2697-2703, 1996.

Langergraber, G. and J. Šimůnek, Modeling variably-saturated water flow and multi-component reactive transport in constructed wetlands, *Vadose Zone Journal*, *4*, 924-938, 2005.

Lenhard, R. J., J. C. Parker, and J. J. Kaluarachchi, Comparing simulated and experimental hysteretic two-phase transient fluid flow phenomena, *Water Resour. Res., 27*(8), 2113-2124, 1991.

Lenhard, R. J., and J. C. Parker, Modeling multiphase fluid hysteresis and comparing results to laboratory investigations, In M. Th. van Genuchten, F. J. Leij, and L. J. Lund (eds.), Proc. Intl. Workshop on *Indirect Methods for Estimating the Hydraulic Properties of Unsaturated Soils,* University of California, Riverside, CA, pp. 233-248, 1992.

Maas, E. V., Crop salt tolerance. In K. K. Tanji (ed.), "*Agricultural Salinity Assessment and Management*", ASCE Manuals and Reports on Engineering practice, No. 71, NY, 1990.

Millington, R. J., and J. M. Quirk, Permeability of porous solids, *Trans. Faraday Soc., 57*, 1200-1207, 1961.

Perrochet, P., and D. Berod, Stability of the standard Crank-Nicolson-Galerkin scheme applied to the diffusion-convection equation: some new insights, *Water Resour. Res., 29*(9), 3291-3297, 1993.

Schaap, M. G., Leij, F. J., and van Genuchten, M. Th., Rosetta: a computer program for estimating soil hydraulic parameters with hierarchical pedotransfer functions, *J. of Hydrol.*, 251, 163-176, 2001.

Scott, P. S., G. J. Farquhar, and N. Kouwen, Hysteresis effects on net infiltration, Advances in Infiltration, *Publ. 11-83*, pp.163-170, Am. Soc. Agri. Eng., St. Joseph, MI, 1983.

Šimůnek, J., and J. W. Hopmans, Parameter Optimization and Nonlinear Fitting, In: *Methods of Soil Analysis, Part 1, Physical Methods,* Chapter 1.7, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 139-157, 2002.

Šimůnek, J., D. Jacques, J. W. Hopmans, M. Inoue, M. Flury, and M. Th. van Genuchten, Solute Transport During Variably-Saturated Flow - Inverse Methods, In: *Methods of Soil Analysis, Part 1, Physical Methods,* Chapter 6.6, Eds. J. H. Dane and G. C. Topp, Third edition, SSSA, Madison, WI, 1435-1449, 2002.

Šimůnek, J., M. Th. van Genuchten, and M. Šejna, The HYDRUS Software Package for Simulating Two- and Three Dimensional Movement of Water, Heat, and Multiple Solutes in Variably-Saturated Media, Version 1.0, Technical Manual, PC Progress, Prague, Czech Republic, 2006.

Stumm, W., and J. J. Morgan, *Aquatic Chemistry: An Introduction Emphasizing Chemical Equilibria in Natural Waters*, John Wiley & Sons, New York, NY, 1981.

Taylor, S. A. and G. M. Ashcroft, *Physical Edaphology*. Freeman and Co., San Francisco, California, p. 434-435, 1972.

van Genuchten, M. Th., A closed-form equation for predicting the hydraulic conductivity of unsaturated soils, *Soil Sci. Soc. Am. J., 44*, 892-898, 1980.

van Genuchten, M. Th., Convective-dispersive transport of solutes involved in sequential first-order decay reactions, *Computers & Geosci., 11*(2), 129-147, 1985.

Vrugt, J. A., J. W. Hopmans, and J. Šimůnek, Calibration of a two-dimensional root water uptake model, *Soil Sci. Soc. Am. J.*, 65(4), 1027-1037, 2001.

Vrugt, J.A., M. T. van Wijk, J. W. Hopmans, and J. Šimůnek, One-, two-, and three-dimensional root water uptake functions for transient modeling, *Water Resour. Res., 37*(10), 2457-2470, 2002.

Vogel, T., and M. Císlerová, On the reliability of unsaturated hydraulic conductivity calculated from the moisture retention curve, *Transport in Porous Media, 3*, 1-15, 1988.

Wesseling, J. G., J. A. Elbers, P. Kabat, and B. J. van den Broek, SWATRE: instructions for input, Internal Note, Winand Staring Centre, Wageningen, the Netherlands, 1991.

Yeh, G. T., and V. S. Tripathi, HYDROGEOCHEM: A coupled model of HYDROlogic transport and GEOCHEMical equilibria in reactive multicomponent systems, *Environs Sci. Div., Publ. No. 3170.*, Oak Ridge National Lab., Oak Ridge, TN, 1990.