

Modelling Variably Saturated Flow with HYDRUS-2D

D. Rassam, J. Šimůnek, and M. Th. van Genuchten

Modelling Variably Saturated Flow with HYDRUS-2D
First Edition

Copyright 2003. Exclusive rights by the publisher 'ND Consult', Brisbane, Australia.

All rights reserved. No part of this book may be used or reproduced in any form or by any means without prior written permission of the publisher.

ISBN: 0-646-42309-6

Contact email: hydrus@optusnet.com.au

Contact fax : + 61 7 3376 7454

Acknowledgment

The lead author would like to acknowledge the efforts of Drs. John Knight and Freeman Cook in helping initiate the project. Special thanks to Dr. Eileen Poeter, Director of IGWMC, for her support.

CONTENTS

Preface	v
Introductory Examples	ix
Example 1: Project ‘1D-Infil’	ix
Example 2: Project ‘2D-infil’	xx
SECTION 1: A Journey Through HYDRUS Windows	
Project Manager	1.1
1.1 <u>Pre-Processing</u>	1.3
1.1.1 Main Processes	1.3
1.1.2 Geometry Information	1.4
1.1.3 Time Information	1.5
1.1.4 Print Information	1.6
1.1.5 Iteration Criteria	1.8
1.1.6 Soil Hydraulic Model	1.10
1.1.7 Water Flow Parameters	1.11
1.1.8 Time Variable Boundary Conditions	1.12
1.1.9 Geometry and Finite Element Mesh Editor	1.13
1.1.9.1 Rectangular Grids	1.13
1.1.9.2 General Grids, MeshGen	1.15
1.1.10 Boundary Conditions Editor	1.26
1.1.10.1 Water Flow Boundary Conditions	1.27
1.1.10.2 Material Distribution	1.35
1.1.10.3 Root Distribution	1.37
1.1.10.4 Initial Conditions	1.37
1.1.10.5 Subregions	1.41
1.1.10.6 Scaling Factors	1.42
1.1.10.7 Observation Nodes	1.43
1.1.10.8 Nodal Recharge	1.43
1.1.10.9 Local Anisotropy	1.43
1.2 <u>Post-Processing</u>	1.44
1.2.1 Graphical Display of Results	1.44
1.2.2 Pressure Heads	1.50
1.2.3 Water Boundary Fluxes	1.50
1.2.4 Cumulative Water Boundary Fluxes	1.50
1.2.5 Soil Hydraulic Properties	1.51
1.2.6 Run Time Information	1.52
1.2.7 Mass Balance Information	1.52
1.2.8 Convert Output to ASCII	1.53

SECTION 2: HYDRUS Output Files

2.1 <u>Output Files</u>	2.1
2.1.1 Boundary.out	2.1
2.1.2 ObsNod.out	2.2
2.1.3 h_mean.out	2.2
2.1.4 v_Mean.out	2.2
2.1.5 Cum_Q.out	2.3
2.1.6 Check.out	2.3
2.1.7 Run_Inf.out	2.4
2.1.8 Balance.out	2.4
2.1.9 Optional Files	2.5
2.1.10 Fit.out	2.5
2.1.11 A_Level.out	2.6
2.2 <u>Examples</u>	2.6
2.2.1 Example Project WT-1	2.6
2.2.2 Example Project Mbal-2	2.8
2.2.3 Example Project Mbal-3	2.12

SECTION 3: Root Water Uptake

3.1 Relevant Windows in HYDRUS	3.1
3.2 Root Water Uptake Simulations	3.3

SECTION 4: Example Applications**Vertical Flow Examples**

4.1 Fluctuating Stream Level	4.1
4.2 Modelling Evaporation	4.3
4.3 Capillary Barrier	4.6
4.4 Modelling Hysteresis	4.8
4.5 Estimating Flow Rates Inside a Domain	4.12
4.6 Effect of Mesh Spatial Discretization and Tolerance Limits	4.16
4.7 Simulating Surface Runoff	4.17
4.8 Local Anisotropy	4.18

Axisymmetric Flow Examples

4.9 Tension Disc Infiltrometer	4.21
4.10 Dewatering of a Cylindrical Pit	4.23

Horizontal Flow Example

4.11 Steady State Horizontal Flow	4.25
-----------------------------------	------

Miscellaneous Examples

4.12 Print Times	4.26
4.13 Running Multiple Sequential Simulations	4.27

SECTION 5: Inverse Solution

5.1 Inverse Solution Options in HYDRUS	5.1
5.2 Inverse Modelling of Controlled Laboratory Column	5.4
5.2.1 Simulating Wetting Cycle 'a'	5.4
5.2.2 Simulating Wetting Cycle 'b'	5.10
5.2.3 Optimising Parameter 'L'	5.14
5.3 Effect of Model's Initial Parameters on Inverse Solution	5.15
5.4 Long Inverse Trial With a Multiple-Layer Soil Profile	5.16
5.4.1 Sensitivity Analysis	5.17
5.4.2 Inverse Simulations With Layered System	5.18
5.5 General Rules; Do's and Don'ts In Inverse Modelling	5.24

SECTION 6: Trouble Shooting

6.1 Infiltration After Long Dry Period	6.1
6.2 Intense Infiltration Into a Two-Layered Soil System	6.4
6.3 Simulations With Highly Unstable Initial Conditions; Drainage of a Saturated Coarse-Grained Soil	6.8
6.4 Infiltration Into a Thick Clay Layer After a Dry Period	6.10
6.5 Simulations With Highly Unstable Initial Conditions; Very High Pressure Gradient	6.15
6.6 Concluding Remarks	6.20

APPENDIXES

<u>Appendix I</u>	Soil Hydraulic Properties	
I.1	Water Retention Parameters	I.1
I.2	Hydraulic Conductivity Function	I.6
I.3	Sensitivity Analysis	I.9
<u>Appendix II</u>	Concepts Related to Modelling Evaporation	
II.1	Significance of Suctions at Soil Surface	II.1
II.2	The Concept of hCritA	II.3
<u>Appendix III</u>	Root Water Uptake	
III.1	Root Water Uptake Reduction Models	III.1
III.2	Root Density	III.3
III.3	Root Distribution Functions	III.4
<u>Appendix IV</u>	Scaling Factors	
IV.1	Background	IV.1
IV.2	Stochastic Scaling Factors	IV.3
<u>Appendix V</u>	Inverse Solution	
V.1	The Inverse Method	V.1
V.2	Statistical Issues	V.2
V.2.1	Distribution of Residuals	V.2
V.2.2	Confidence Limits	V.4
V.2.3	Goodness of Fit and Errors	V.5
V.2.4	Correlation Matrix	V.6
V.2.5	Weighting Parameters	V.7
V.3	Intelligent Data Acquisition	V.9
<u>Appendix VI</u>	Introductory Examples on Solute Transport	VI.1
<u>Appendix VII</u>	Index of HYDRUS Windows	VII.1
<u>Appendix VIII</u>	Description of Toolbars	VIII.1
REFERENCES		Ref.1
ALPHABETICAL INDEX		Index.1

PREFACE

This document is a user manual for the HYDRUS-2D software package (Šimůnek et al., 1999), which simulates water flow and solute transport in two-dimensional variably saturated media. The manual mostly relates to modelling water flow; however, several introductory examples on solute transport are discussed in an appendix. Over one hundred example projects are included on the accompanying CD. Most of the example projects on the accompanying CD were prepared using HYDRUS-2D version 2.007; hence you might encounter problems when using older versions of the software.

This manual covers in details all aspects of modelling water flow that can be accomplished with HYDRUS-2D. It includes step-by-step procedures for beginners, as well as techniques and tips for advanced users. Many of the example applications and tips were inspired by numerous questions and comments put forward by users through the HYDRUS discussion group at www.pc-progress.cz.

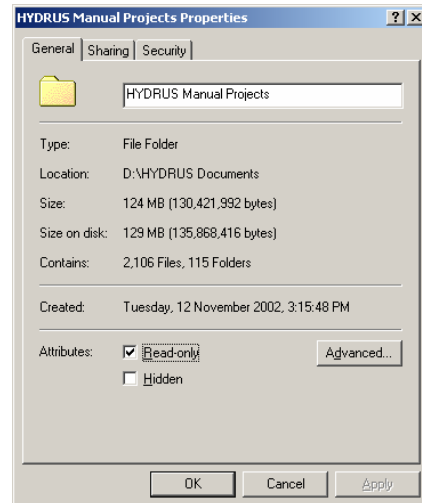
This manual is organised into eight major parts:

- **Introductory Examples**: Includes two basic examples designed to demonstrate the capabilities of the software for first-time users.
- **Section 1**: This section takes the user on a journey through all HYDRUS windows. It covers various aspects that include starting new projects, setting up finite element grids, assigning boundary conditions, running the model, and viewing the results.
- **Section 2**: This section explains the various output files that HYDRUS produces. It explains the meaning of the different variables involved, and provides examples on how to manipulate the output data.
- **Section 3**: This section includes example projects on root water uptake.
- **Section 4**: This section includes 13 example applications. Examples 1-8 are ‘Vertical Plane Flow’ examples. Examples 9 and 10 are ‘Axisymmetric Vertical Flow’ examples. Example 11 is a ‘Horizontal Plane Flow’ example. Examples 12 and 13 show extra features for advanced users.
- **Section 5**: This is a comprehensive section on inverse parameter estimation. It demonstrates through numerous examples important issues such as non-uniqueness of a solution, effects of temporal and spatial distributions of the calibration data, effects of initial values of the optimised parameters, statistical issues, and validation of results. This section ends with a list of recommendations on inverse modelling.
- **Section 6**: This section relates to trouble-shooting. We demonstrate through many examples cases where HYDRUS may not perform as well as it should, present likely reasons, and put forward measures for avoiding such problems.
- **Appendixes**: In Appendixes I to V we present theoretical backgrounds related to soil hydraulic properties, modelling evaporation and root water uptake, the meaning of scaling factors, and various aspects related to inverse parameter estimation. Appendix VI presents two basic examples on modelling solute transport. Appendix VII includes an alphabetical index of HYDRUS windows with cross-referencing to related pages and examples in the manual. Finally, Appendix VIII includes a list of available toolbars and their meaning.

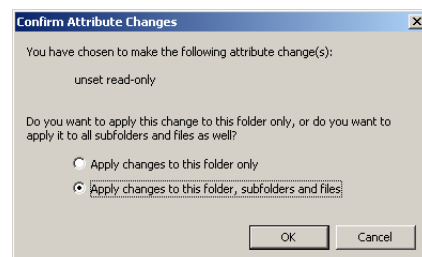
Example projects on the CD

Follow the instructions below if you want to use the ‘HYDRUS Project Manager’ to access the example projects shipped on the CD that accompanies the manual:

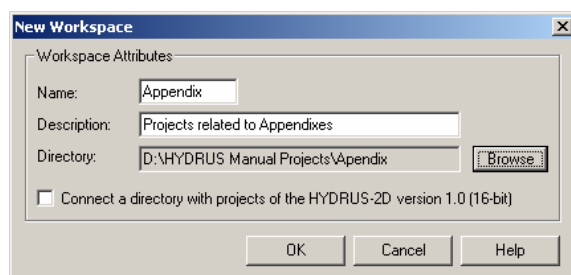
1. Copy directory ‘HYDRUS Manual Projects’ from the CD to your local hard drive.
2. In ‘Windows Explorer’, right click on the directory and select ‘Properties’; this window will then appear.
3. Uncheck the ‘Read only’ box and hit OK.



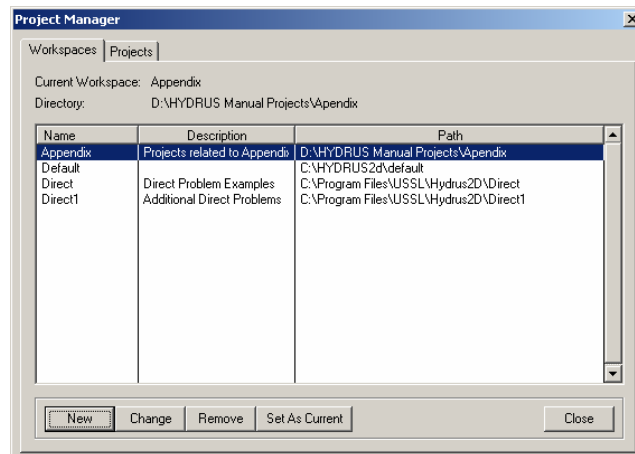
4. You will be prompted to confirm the attribute changes. Check the lower box ‘Apply changes to this folder, subfolders and files’, and hit OK. This will allow HYDRUS to access the project files.



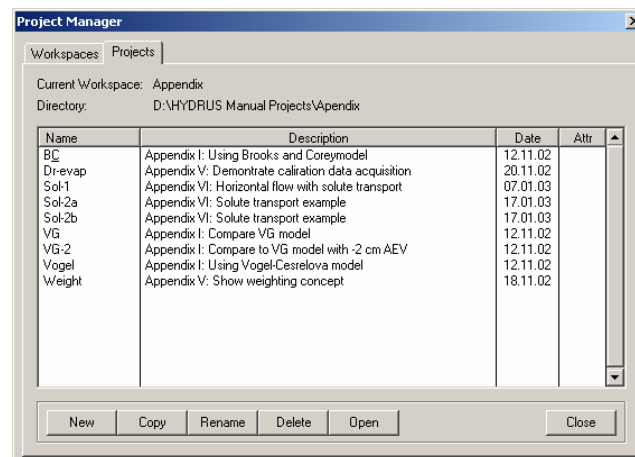
5. Since these projects were copied from an external source, you will need to inform HYDRUS that they exist so that you can access them from ‘Project Manager’. In this example, we are setting a workspace for the projects saved in the ‘Appendix’ subdirectory (where all projects related to the appendixes are stored). Start HYDRUS, ‘File/Project Manager/Workspaces/New’, enter the information as shown here (in Browse, you should provide the relevant directory path on your computer). Hit OK. Note that ‘D:\.’ here refers to the path on your hard drive.



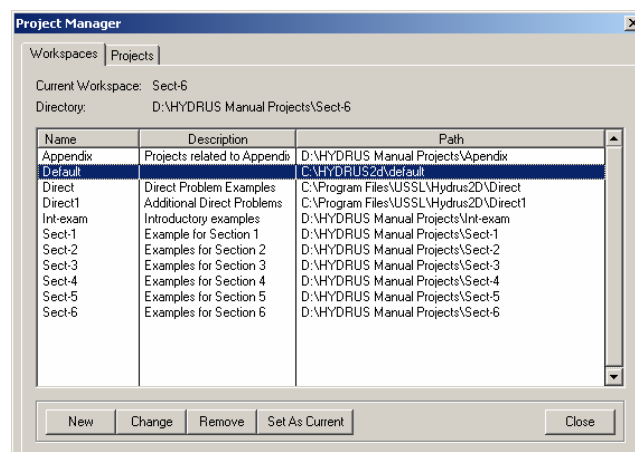
6. You will see that the ‘Workspaces’ window will now look like this.



7. Double click on the ‘Appendix’ Workspace (or select ‘Appendix’ and click on ‘Projects’). The ‘Projects’ window should look like this (you should have 9 projects in this workspace).



8. Repeat this process 7 times, each time providing the relevant directory structure. The final ‘Workspaces’ window should look like this.

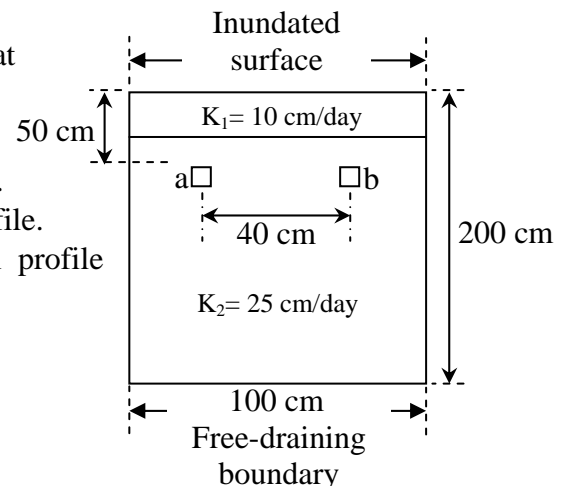


INTRODUCTORY EXAMPLES

In this introduction, we will present two water-flow examples that will cover some of the basic features of HYDRUS-2D. The examples are meant to demonstrate the basic capabilities of the software package and to guide first-time users.

The examples represent a 200 cm-deep soil block having two horizons (the top 30 cm has a lower hydraulic conductivity). The soil initially has a water content of 0.1. The water table is located far below the freely draining bottom boundary. We model 5 days of surface saturation. The simulation will provide the following information:

- How the water content of the soil changes at two locations, 'a' and 'b' in the profile
- Advance of wetting boundary after 1 day.
- How much water infiltrated the soil profile.
- How much water drained from the soil profile.
- How much water was stored in the soil profile during the 5-day period.



Example 1: 1-Dimensional infiltration problem.

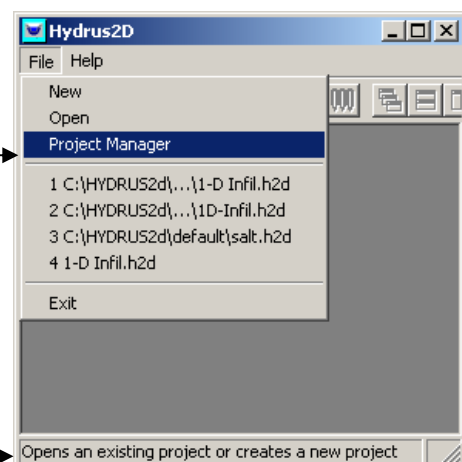
Example 2: 2-Dimensional infiltration problem. This example is almost identical to the first; we convert example 1 into a 2-dimensional problem by simply changing the width over which infiltration is applied.

1. Project '1D-Infil'

We will create this project step-by-step. The final project is found on the CD (**..\HYDRUS Manual Projects\Int-exam**).

Start HYDRUS-2D
File, Project Manager 

*Note that as you move the mouse over 'Project Manager', the dynamic help appears in the window explaining what it does.

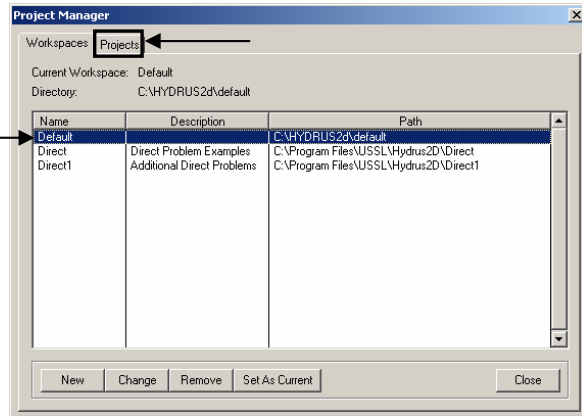


This is the ‘Project Manager’ window.

Select the first line ‘Default’; this is the directory where the project will be saved.

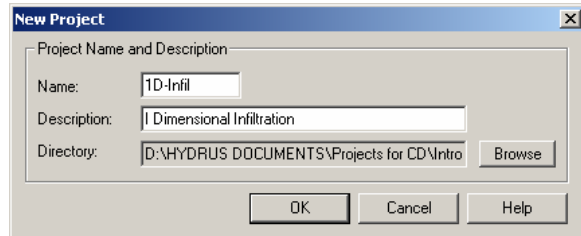
Click ‘Projects’

A new window will appear with a list of projects saved in this workspace; if you are trying this for the first time, this workspace will be empty. We will now create a new project in this workspace.

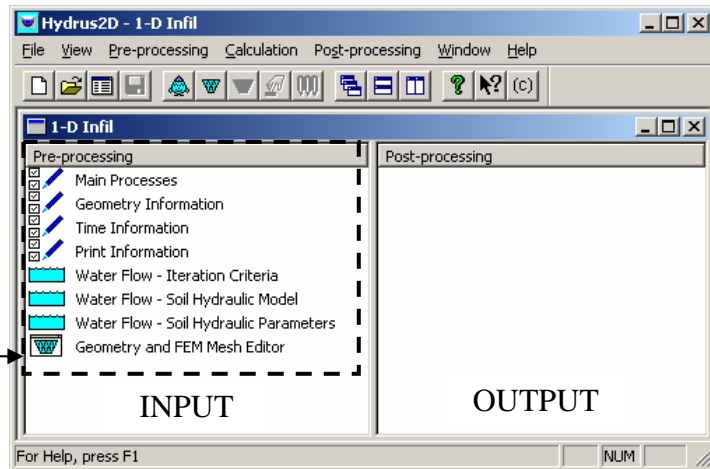


In the ‘Projects’ window, select ‘New’

Enter the details as shown here.
Select OK.



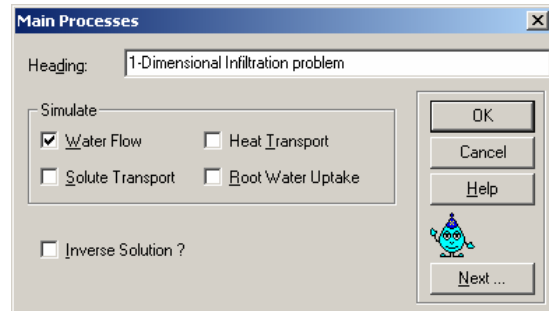
This window will appear. This is the main HYDRUS window, through which you can access all available functionalities in the software. Alternatively, you can access them also from the menu.



The easiest way to set a new project is to go step-by-step through the Pre-Processing section shown above.

Double-Click ‘Main Processes’

Enter the Heading section and check the box Water Flow.



Next....

Enter the information as shown here:

- We are using centimetres for length units.
- We are modelling flow in a vertical plane.
- We are using a basic rectangular geometry.
- We have 2 materials.
- We want 2 mass balance regions (more relevant to next example).

Geometry Information

Length Units: mm cm m

Type of Flow: Horizontal Plane Vertical Plane Axisymmetrical Vertical Flow

Geometry Type: Rectangular General

Soil Profile: 2 Number of Materials (Heterogeneity) 2 Number of Layers (Mass Balances)

Buttons: OK, Cancel, Help, Next ..., Previous ...

Next....

Enter the information as shown here:

- We are using days for time units.
- The simulation length is 5 days.
- The remaining numbers relate to the numerical time-stepping scheme; they are explained in Section 1.1.3.

Time Information

Time Units: Seconds Minutes Hours Days

Time Discretization: Initial Time 0, Final Time 5, Initial Time Step 0.001, Minimum Time Step 0.0001, Maximum Time Step 1

Boundary Conditions: Time-Variable Boundary Conditions, 0 Number of Time-Variable Boundary Records

Buttons: OK, Cancel, Help, Next ..., Previous ...

Next....

Enter 10 for number of print times

Click 'Select Print Times'

Print Information

Print Options: I-Level Information, Screen Output, Print Fluxes

Number of Print Times: 10

Buttons: OK, Cancel, Help, Next ..., Previous ...

Select 'Print Times', in this case click 'Default' for uniform spacing of print times ($5/10=0.5$ day).

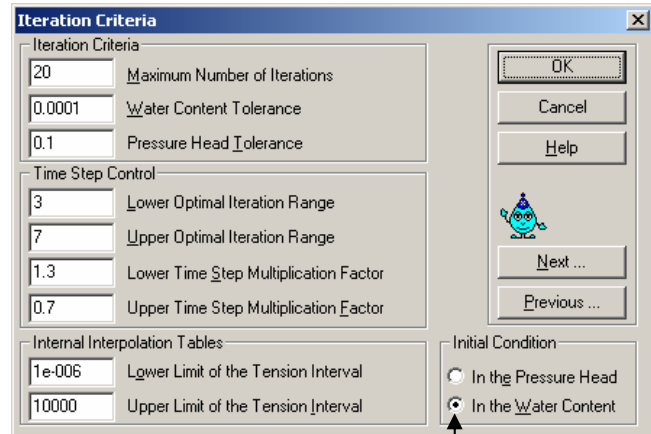
	1	2	3	4	5
Print Times	0.5	1	1.5	2	2.5

Buttons: Default, OK, Cancel

OK....Next....

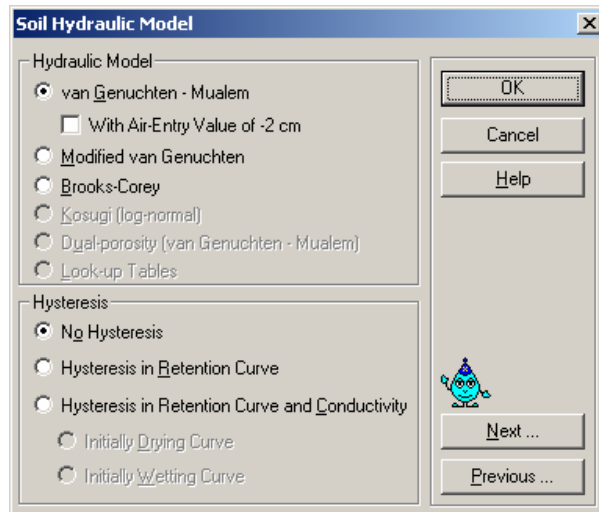
Leave all default values in this window except ‘Initial Conditions’.
Details on the remaining parameters are found in Section 1.1.5.

Check the radio button “In the Water Content” as shown here. This means that our initial conditions will be interpreted as water contents rather than pressure heads.



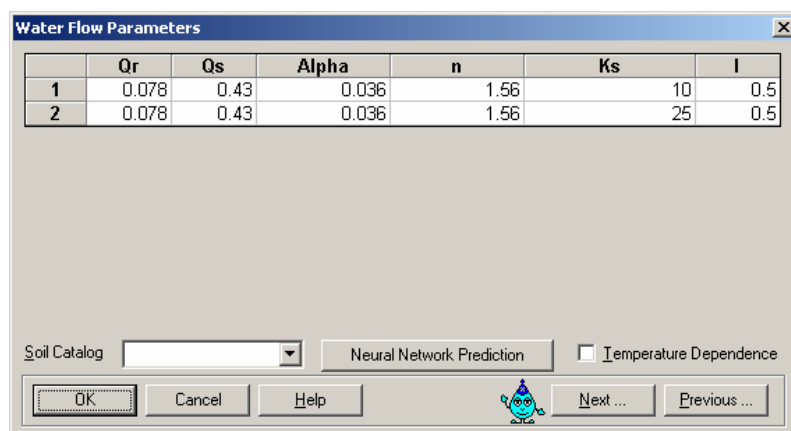
Next....

Leave the default model as shown here.
We use the van Genuchten-Mualem model to describe the soil hydraulic properties.



Next....

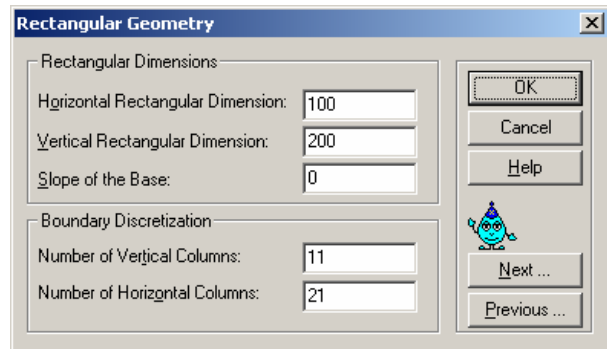
Enter the following water retention parameters. Note that the row number here refers to the material number (two materials in our case).



Next....

Enter the values as shown here.

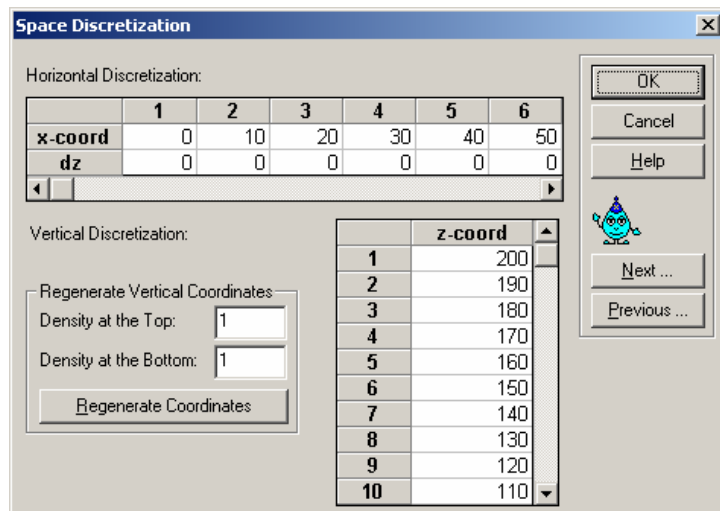
- Horizontal rectangular dimension (width) is 100 cm.
- Vertical rectangular dimension (depth) is 200 cm.
- We chose to split (discretize) the width into 11 columns (10 cells).
- We chose to split (discretize) the depth into 21 columns (20 cells or elements).



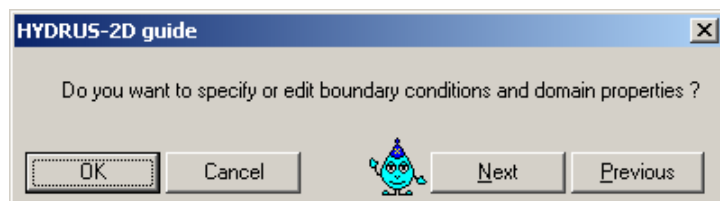
Next....

Accept the default uniform grid shown in this window.

The discretization resulted in a uniform 10-cm square grid system.



Next....



OK....

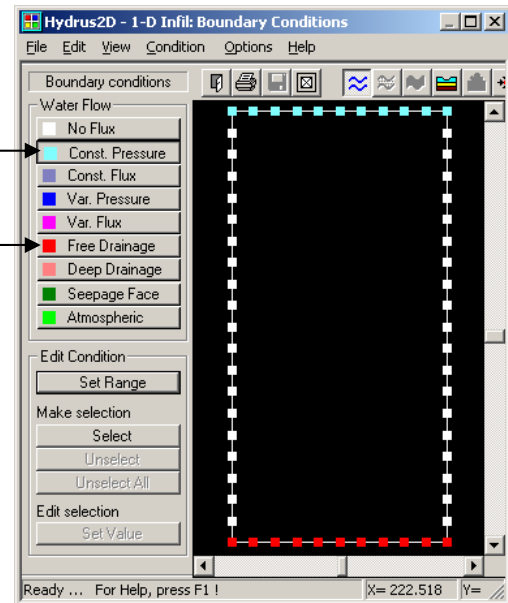
Assign Water Flow Boundary Conditions



We will assign a Constant Pressure boundary conditions (BC) to the surface nodes and a Free Drainage BC to the lower nodes.

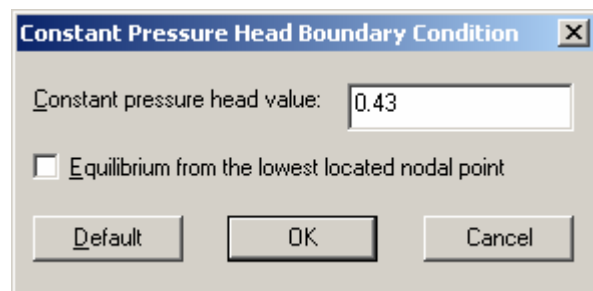
Click the Const. Pressure button, and notice the change in the cursor.

Left-click and drag the cursor over the top nodes, and then left-click again to end. The following window will appear.



Enter the value 0.43, which corresponds to saturated water content conditions. Although the window prompts for constant pressure, remember that we opted to enter initial conditions as 'In the Water Content'.

Click OK.



Click on Free Drainage, and repeat the same for the lower nodes. There is no value associated with this BC.

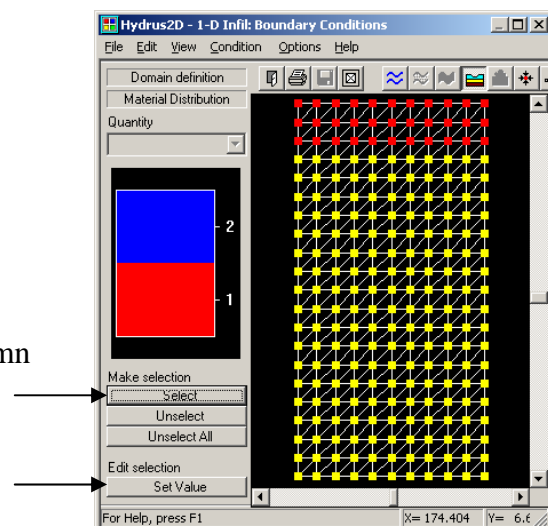
Assign Material Distribution:

Condition, Material Distribution

Click Select (from sidebar), the cursor will change.


Left-click slightly to the left of the first column and above the fourth row and drag below the bottom row and last column to select all nodes below row 3.

Click Set Value (sidebar buttons)



A window will appear, enter number 2. This means that all selected nodes below the third row are assigned material number 2 (which corresponds to the row numbers in the window 'Water Flow Parameters'). The top 3 rows default to material number 1.

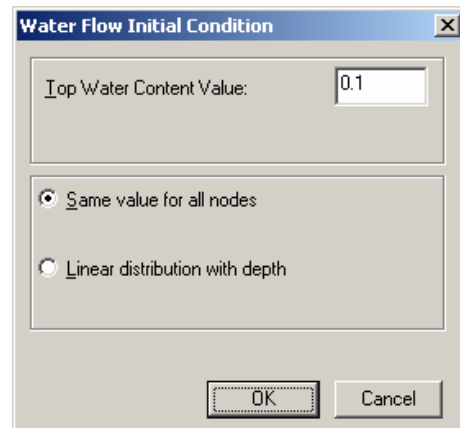
Assign Initial Conditions:

Conditions/Initial Conditions ()
Pressure Head/Water Content


Click Select (sidebar button as done previously in Material Distribution).

Select all nodes from row 2 right down to the bottom boundary. Note that we exclude the first row because it is a constant head BC and its value has already been prescribed.

Click Set Value (sidebar button), and this window will appear. Check 'Same value for all nodes', and enter the initial water content 0.1

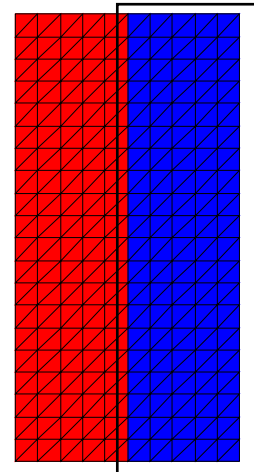


Assign Subregions:

Condition/Subregions () sets the number of mass balance regions. Remember that we earlier decided on 2 in the "Geometry Information" window.

Click Select (sidebar button); select the nodes as shown here.

Click Set Value (sidebar button); enter the number 2 meaning that the right-hand side rectangle will be subregion 2.

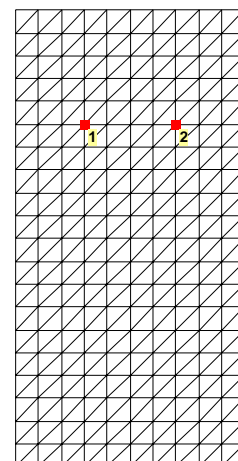


Assign Observation Nodes:

Condition/Observation Nodes ()

Click Insert (sidebar buttons); cursor changes to open square.

Select the nodes as shown here.



Note: If you fail to enter the right number of materials or subregions then a warning will be issued in this window and you will be requested to do so.

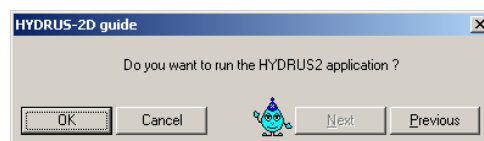
Click the save button, or File/Close, and you will be prompted to save.

Running the simulation

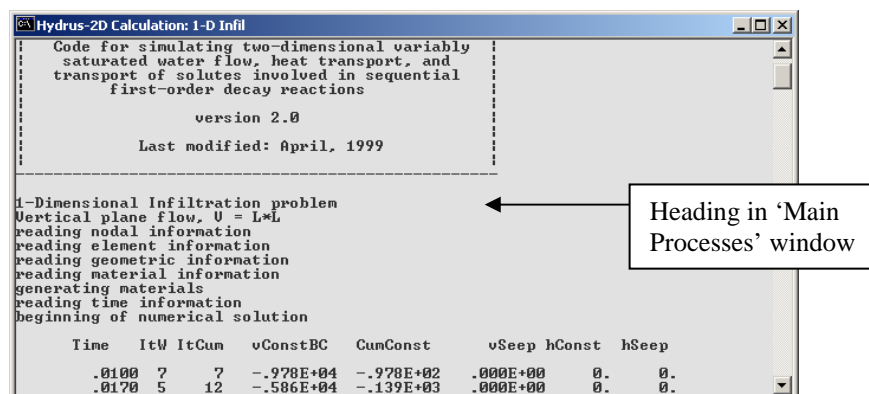
Click, Calculation, Run HYDRUS-2D ().

Click OK to run.

When Screen Output is checked in the 'Print Information' window, the results will be dynamically displayed as the simulation proceeds.



For our current problem, this DOS window provides the following information:



Column 1	Time
Column 2	Number of iterations at each time step
Column 3	Cumulative number of iterations
Column 4	Constant head/saturation BC flux
Column 5	Cumulative volume for constant head/saturation/flux BC. Flux across boundary with prescribed constant head/saturation/flux BC
Column 6	Flux across seepage face (not assigned in this example)
Column 7	Average pressure head at constant head/saturation BC
Column 8	Average pressure head at seepage face (not assigned in this example)

At the end of the simulation, hit the Enter keyboard button to return to the HYDRUS window (you can always hit the Enter button before the simulation ends).

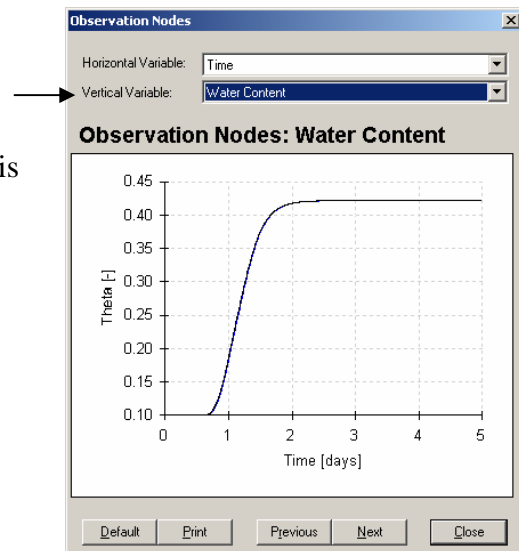
Display the results in the Post-processing window

- How the water content of the soil changes with time at two locations (observation nodes) '1' and '2' in the profile (on the previous page).

In Post-processing part (right side of the HYDRUS main window), double-click on Observation Points

In Vertical Variable list box, choose Water Content and this figure will show.

Notice that the two observation points produced identical results. Although this problem is run as a 2-dimensional domain, it is basically a 1-dimensional problem. That is, results are the same at any point along a horizontal line across the domain.



Click Close.

- Advance of wetting boundary after 1 day

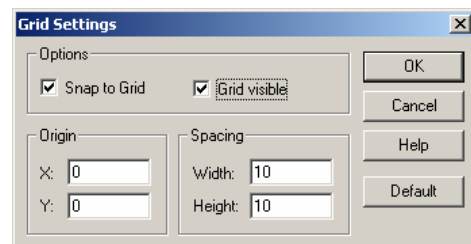
Double-click on Graphical Display of Results (📊)

On the sidebar buttons, scroll the Time-value bar to 1 (day).

Quantity/Water Content

Options/Grid Settings (⚙️), enter the values as shown.

Click OK.

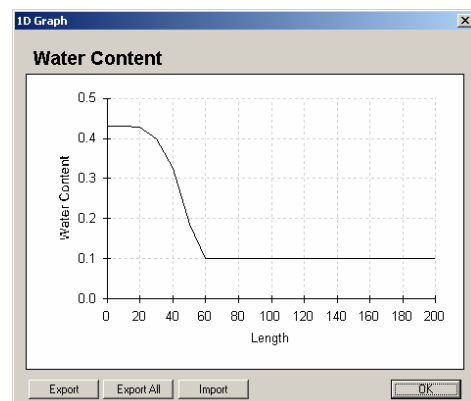


1 D Graph/Cross Section (📐)

Select a node on the upper boundary (anywhere since results do not vary horizontally), move the mouse vertically and click on the lower boundary. This graph showing the water content profile after 1 day will appear (notice the shape of the wetting front).

Click OK.

File/Close; Return to the main Hydrus-2D module.

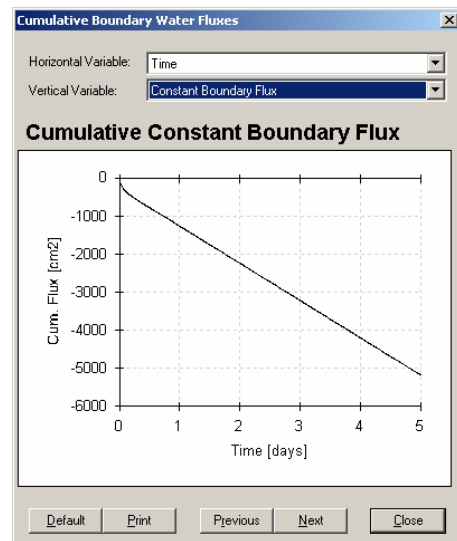


- How much water infiltrated into the profile?

Double-click on Cumulative Water Boundary Fluxes.

For the Vertical Variable, select Constant Boundary Flux.

Notice that fluxes into the transport domain are negative.



Right-click on the chart
Select Edit Chart Data

Scroll down until C1 reads 5, which is the final time (5 days). The volume of water that has infiltrated from the constant-head boundary (soil surface) is 5190 cm^2 (cm^3/cm); the negative sign indicates water entering the domain.

Click OK (leave the Cumulative Water Boundary Fluxes dialog open).

	C1	C2
R663	4.9531	-5140
R664	4.9609	-5150
R665	4.9688	-5150
R666	4.9766	-5160
R667	4.9844	-5170
R668	4.9922	-5180
R669	5	-5190

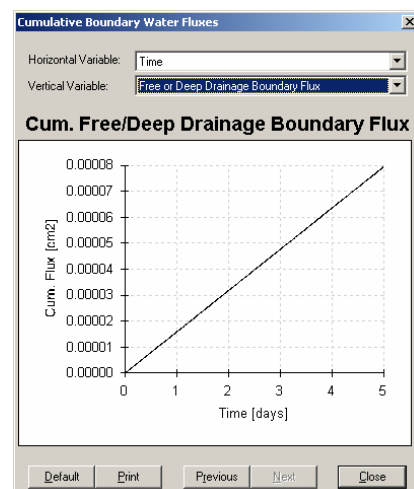
- How much water drained from the profile?

As Vertical Variable, select Free or Deep Drainage Boundary Flux

This window shows that drainage from the lower boundary is almost zero.

Notice that fluxes out of the transport domain are positive.

Click Close.



- How much water was stored in the profile.

HYDRUS keeps track of the mass balance and provides results at print times.

Double-click on Mass Balance Information.

We are looking here at the total mass balance, that is, for the whole region regardless of subregions.

Time [T]	Total	Sub-region number ...
.0000		1 2
Area [V]	.200E+05	.100E+05 .100E+05
Volume [V]	.217E+04	.108E+04 .108E+04
InFlow [V/T]	.000E+00	.000E+00 .000E+00
hMean [L]	-.383E+04	-3826.4 -3826.4
.5000		1 2
Area [V]	.200E+05	.100E+05 .100E+05
Volume [V]	.295E+04	.148E+04 .147E+04
InFlow [V/T]	.944E+03	.474E+03 .470E+03
hMean [L]	-.312E+04	-3108.3 -3126.9

At time 0, Volume = .217E+4 (volume of water stored in the soil)

Scroll down to the final time, as shown here.

At time 5, Volume = .735E+4

hMean [L]	-.103E+04	-1017.8 -1042.8
WatBalT [V]	-.127E+01	
WatBalR [%]	.030	
4.5000		1 2
Area [V]	.200E+05	.100E+05 .100E+05
Volume [V]	.686E+04	.343E+04 .343E+04
InFlow [V/T]	.979E+03	.489E+03 .490E+03
hMean [L]	-.721E+03	-717.3 -725.1
WatBalT [V]	-.142E+01	
WatBalR [%]	.030	
5.0000		1 2
Area [V]	.200E+05	.100E+05 .100E+05
Volume [V]	.735E+04	.368E+04 .367E+04
InFlow [V/T]	.979E+03	.490E+03 .490E+03
hMean [L]	-.429E+03	-416.9 -440.9
WatBalT [V]	-.160E+01	
WatBalR [%]	.031	

Therefore, the change in storage= $7350-2170=5180$ (gain, increase in storage).

This value is very close to the cumulative infiltration (5190). The difference is the error in the mass balance (remember that free drainage is almost zero).

2. Project '2D-Infil'

This project is found on the CD (..\HYDRUS Manual Projects\Int-exam)

Project Manager (📁)

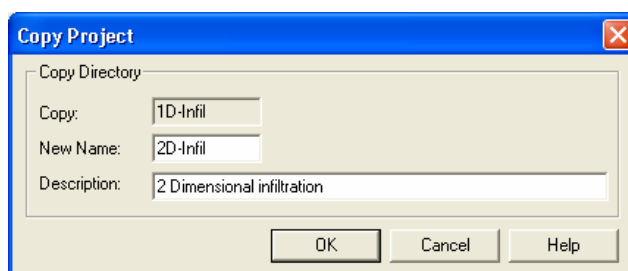
Select Project '1D-Infil'

Copy

Enter the name as shown

OK

Open



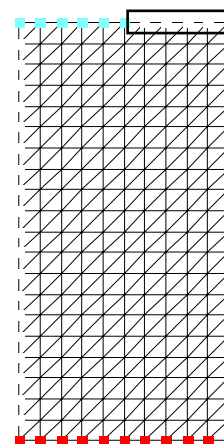
This will create an identical copy of the previous project.

Double click on Boundary Condition Editor (📏)

Select No Flux, from the sidebar buttons

Left-click and drag over the right hand surface nodes as shown here.

Left-click again to end.



Conditions/Initial Conditions/
Pressure Head/Water Content

Click Select

Select the same surface nodes (shown in figure here) to change the initial condition from 0.43 (specified here as a boundary condition for the previous run)

Set the initial water content to 0.1

Save the changes; a warning will appear that the existing output files will be deleted, click OK.

Calculation/Run HYDRUS-2D/OK (🏠)

View results:

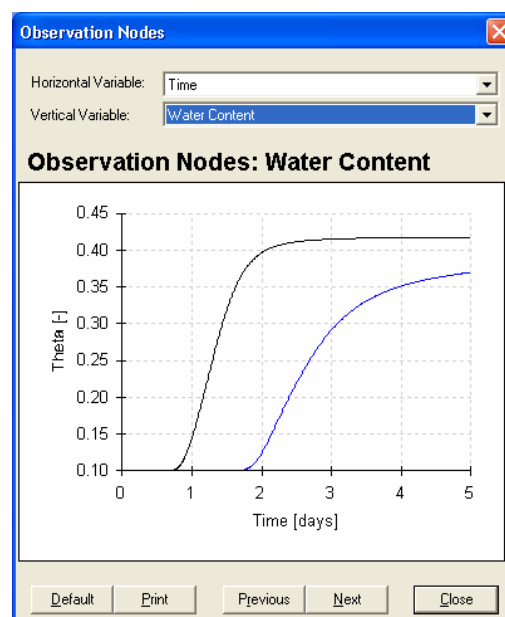
- How the water content of the soil changes at two locations 'a' and 'b' in the profile

In Post-processing, double-click on


Observation Points.

As Vertical Variable, choose Water Content.

We see here that the response is now different for the two observation points. The problem is not 1-dimensional any more.



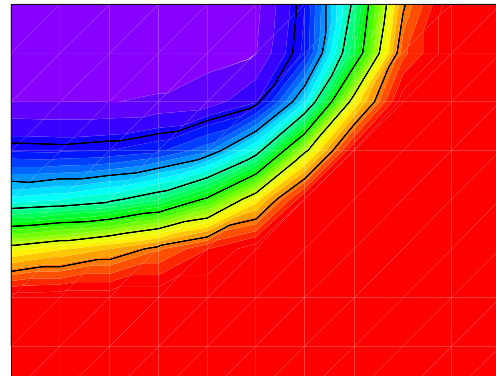
- Advance of wetting boundary

Double-click on Graphical Display of Results
()

On the sidebar buttons, slide the Time-value scroll bar to 1 (day).

Quantity/Water Content

2-D Graph/Spectrum +Isoline ()



The figure here shows that the wetting boundary advances in a 2-dimensional manner.

- How much water infiltrated the profile?

Repeat the same procedure as before to access the cumulative constant boundary flux; it is equal to 3000 cm^2 (much less than before).

- How much water drained from the profile?

Free drainage is still close to zero.

File/Close - Close the Graphics module

- How much water was stored in the profile during the 5-day period?

Click on Mass Balance Information

Time [T]	Total	Sub-region number ...	
		1	2
.0000			
Area [W]	-.200E+05	.100E+05	.100E+05
Volume [V]	-.209E+04	.108E+04	.101E+04
InFlow [V/T]	.000E+00	.000E+00	.000E+00
hMean [L]	-.387E+04	-3826.4	-3918.0
.5000			
Area [W]	.200E+05	.100E+05	.100E+05
Volume [V]	.256E+04	.146E+04	.110E+04
InFlow [V/T]	.567E+03	.425E+03	.142E+03
hMean [L]	-.342E+04	-3148.9	-3688.7

At time 0

Total volume = $1080 + 1010 = 2090 \text{ cm}^3$ (sum for the two regions).

Notice that the initial volume in region 1 is slightly higher because the surface constant head BC nodes were saturated.

At time =5 days

Total volume
 $=3000+2080=5090 \text{ cm}^3$

Notice that region 1 (below
 the constant head BC)
 contains more water.

Mass Balance Information			
hMean [L]	-.193E+04	-1556.0	-2306.4
WatBalT [V]	-.110E+01		
WatBalR [%]	.045		
4.5000			
Area [V]	.200E+05	.100E+05	.100E+05
Volume [V]	.481E+04	.286E+04	.195E+04
InFlow [V/T]	.561E+03	.294E+03	.267E+03
hMean [L]	-.174E+04	-1365.9	-2110.1
WatBalT [V]	-.120E+01		
WatBalR [%]	.044		
5.0000			
Area [V]	.200E+05	.100E+05	.100E+05
Volume [V]	.509E+04	.300E+04	.208E+04
InFlow [V/T]	.561E+03	.286E+03	.274E+03
hMean [L]	-.155E+04	-1180.9	-1923.4
WatBalT [V]	-.128E+01		
WatBalR [%]	.043		

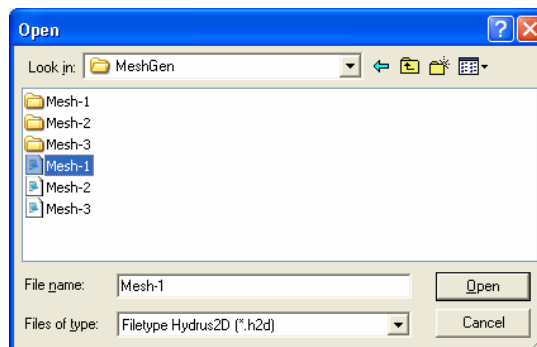
The increase in storage during the 5-day period= $5090-2090=3000 \text{ cm}^3$ (equal to the constant head BC cumulative flux).

1. A JOURNEY THROUGH HYDRUS WINDOWS

Opening Existing HYDRUS Projects

File/Open:

This window will appear. The file type that HYDRUS looks for has the extension “.h2d”. Each project has such a file that is located in its parent directory. In the window shown here, directory “MeshGen” contains three projects, Mesh-1, 2, and 3. Notice that there are three “*.h2d” files that have names identical to the directories where the projects are stored (the directory name is identical to the project name). The *.h2d file is not project-specific, that is, it does not relate to the contents of the project in any way. Hence, it is merely a project name file. It is possible to copy this file, re-name, and then use it with another project.

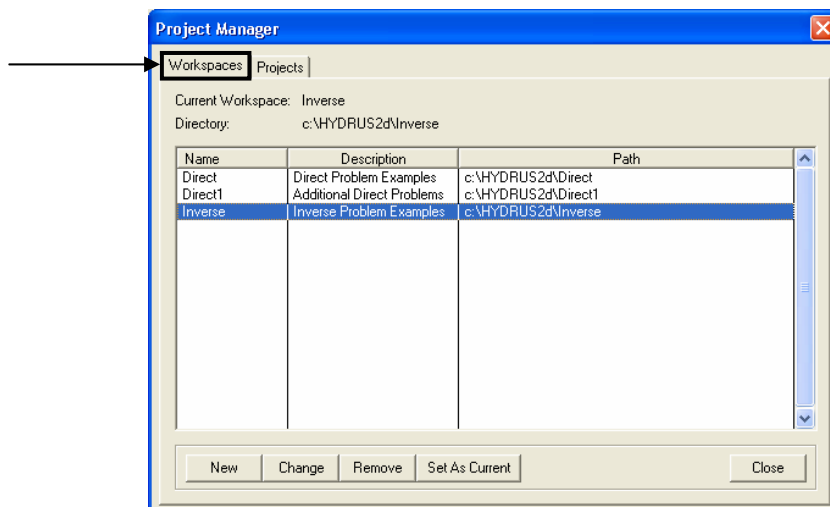


Alternatively, and much easier, a project can be opened directly from the Project Manager.

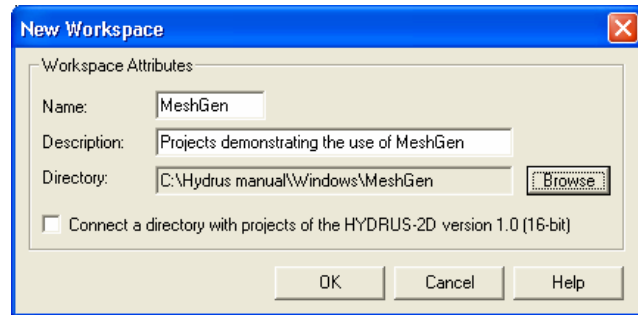
Project Manager

Workspaces:

Workspaces refer to directories that contain HYDRUS projects. Each workspace is characterized by its name, brief description and path. The window here shows the default workspaces that contain the project examples shipped with the software.



New: Will set a new workspace. Enter a name, provide a description, and browse to specify a path.



Change: Select an existing workspace and click “change”. A similar window will then appear, which allows changes to be made to the details of the selected workspace.

Remove: Will remove a workspace from the list (will not delete existing projects).

Set as current: Select the workspace that you will work in. The projects in this workspace will appear whenever you open Project Manager.

Projects:

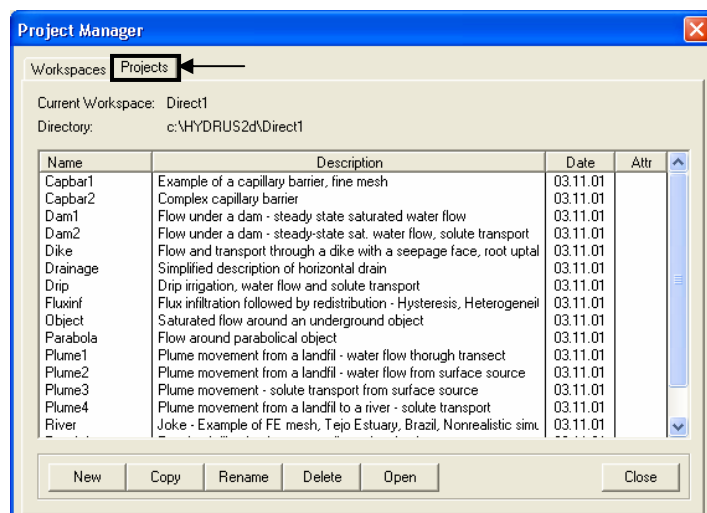
Projects refer to a particular HYDRUS run. Its input and output files are stored in one subdirectory. A project is characterized by its name (up to 8 characters) and a brief description. Select a workspace, then click on the Projects button on top of the Project Manager Window.

New: Creates a new project

Copy: Copies projects. You should provide a new name for the copied project.

Rename: Change any of the details of the project, like its name or description.

Remove: This option deletes projects; it is irreversible and hence gives you a warning.



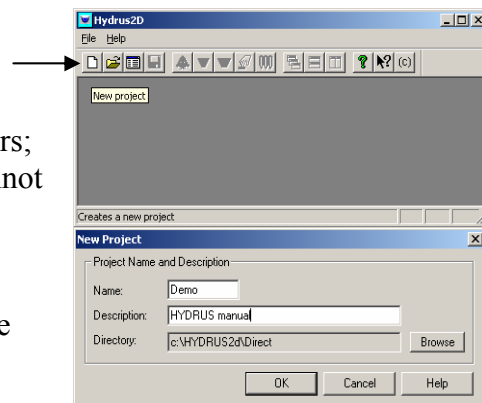
Open: Select a project from the list and click “open” to work on that project. Alternatively, projects may be opened also by double-clicking on its name or description.

We recommend that project manipulations be carried out within the Project Manager rather than using Windows Explorer or the File/Open command.


Start New Project

Click the “New Project Icon” () , or go to File/Project Manager/Projects/New

- Enter the project name (use up to 8 characters; longer names are possible, but then projects cannot be manipulated using Project Manager).
- Enter a brief description.
- Specify a directory where the project is to be saved.

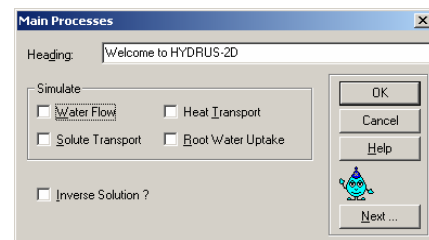


1.1 Pre-Processing

- For a new project, we recommend that you go sequentially through the windows. Start by double clicking on “Main Processes” (or ) , and then click button “next” to go to subsequent windows.
- If changes are to be made to an existing project, you can go directly to the relevant window.

1.1.1 Main Processes

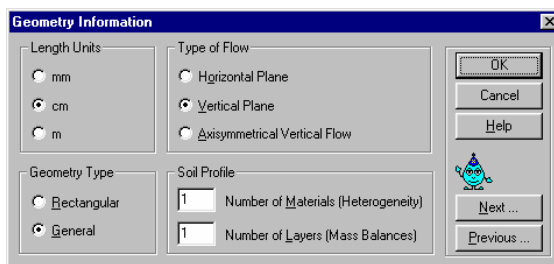
- Heading: Type a heading that you want to appear in the output files.
- Simulate: Specify the scope of your problem. You can simulate water flow, solute transport, root water uptake, heat transport, or any combination of these processes. The number of options (or windows) available under “pre-processing” will depend upon the options that you select in this window. Deselecting any of these options in existing projects causes you to lose all data relevant to the unchecked option.
- Inverse Solution: For normal simulations or “forward problems”, you provide the known input parameters and HYDRUS will run once through the specified time duration. However, under the “inverse solution” option, HYDRUS will compare the simulation results with observed experimental data (that you provide), and then re-run the model a number of times (the number depends on how correct the initial input parameters were) with a new set of parameters until a close fit to the observed experimental data is obtained. This process is alternatively referred to as “Model calibration”, or “Inverse parameter estimation”. Refer to Section 5 and Appendix V for more details.
- Note that unchecking all boxes will result in a steady-state analysis of water flow for specified boundary conditions.



Click next....

1.1.2 Geometry Information

- **Length Units:** Selects the length units. You have to be consistent throughout the project. We will be using mostly “cm” throughout this document. Note that any later change in units will lead to conversion of the entire input data set to new units. We do not recommend this action without additional checking of consistency of input data.



- **Geometry Type:** The “Rectangular” option defines a simple structured finite element mesh. The “General” option, which requires the MESHGEN2D interface, can handle more complex geometries (see Figure 1.1).

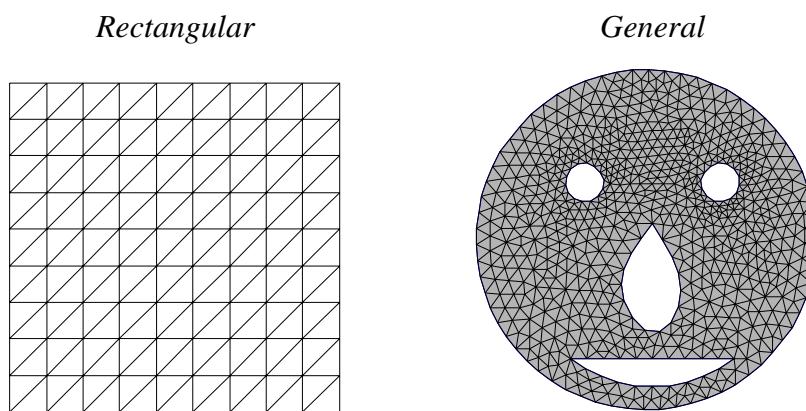


Figure 1.1: Rectangular and General meshes

Type of Flow:

- **Vertical Flow** models an infinite soil block in the y-direction as demonstrated in Figure 1.2a.

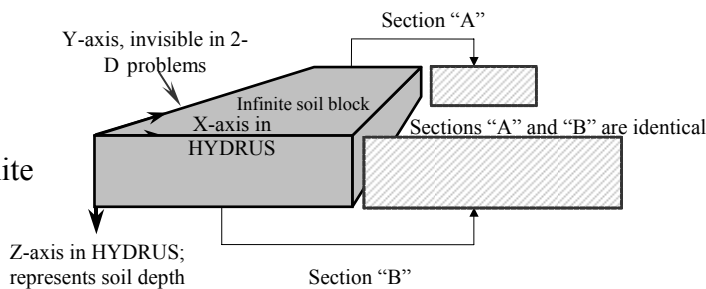


Figure 1.2a: 2-Dimensional vertical plane flow

- **Axisymmetric Flow** considers radially finite, 3-dimensional geometry as shown in Figure 1.2b where the modelled 2-D plane represents a 3-D cylindrical shape.

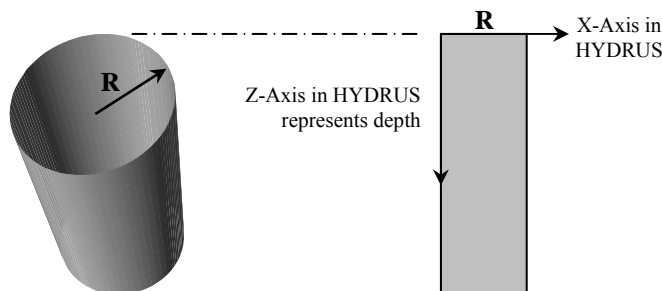


Figure 1.2b: 3-Dimensional axisymmetric flow

- Horizontal Flow is modelled in the x-y plane that ignores the effect of gravity (see x-y plane in Figure 1.2a). Refer to example 4.11.

Soil Profile:

- Number of Materials: You can introduce heterogeneity by selecting more than one material. You'll be prompted later to enter several sets of soil hydraulic or other parameters, equal to the material number you enter here. In addition, you'll have to define where these materials are located within the domain (in the "Boundary Conditions Editor").

- Number of Layers (mass balance sub-regions) does not affect the solution but provides more detailed mass balance calculations relevant to particular regions of interest that you will need to specify later. If you leave the default value of 1, HYDRUS will provide at each print time the mass balance calculation only for the entire domain.

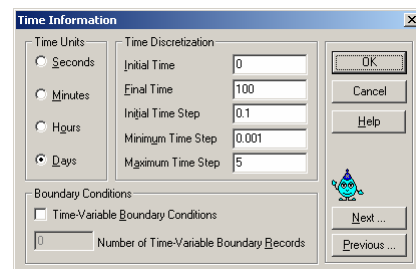
For example, in a two-dimensional drainage problem you might like to know how the block next to the drain is draining compared to another part in the domain that is away from the drain.

Under the section "Boundary conditions", you'll have to define later a number of **sub-regions** equal to the number of mass balance layers that you entered in this window. If you define a different number of sub-regions, a warning will be issued and you will be requested to enter the correct number.

Click next....

1.1.3 Time Information

- Time Units: Select the time unit. You have to be consistent throughout the project. HYDRUS re-adjusts all time-related variables if the unit system is changed after setting the model.
- Time Discretization: Enter initial and final times. Initial time is usually zero, except for more advanced applications when re-starting existing runs (see Section 4.1).



Selection of the initial time step relates to the numerical solution, which is a self-adjusting time marching scheme. This is the initial time step that HYDRUS adopts at the beginning of the solution and whenever boundary conditions change significantly. As the iterative numerical solution finds it more difficult to converge, the time step is automatically reduced. However, a limit is introduced on how small the time step is allowed to become. This limit is the minimum time step. We recommend allowing the minimum time step to be on the order of 1 s.

On the other hand, if the solution is converging fast, the time step is increased. The maximum time step is a limit on how large the time step can become.

More details on the time marching scheme are provided in Section 1.1.5 and Chart 1.1.

- Time-Variable Boundary Condition allows a user to include atmospheric data such as precipitation and evaporation, plant transpiration, and time variable BCs such as pressure heads and/or fluxes; the relevant data is input as a time-series. Once this option is selected, the box ‘Number of Time-Variable Boundary Records’ will be activated and prompt the user to enter an integer ≥ 1 .

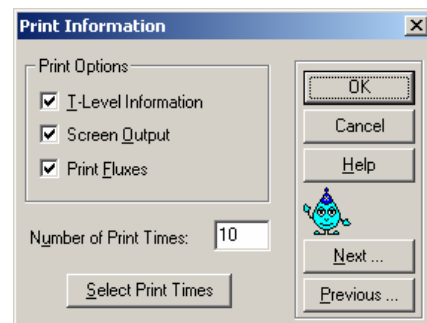
For example, when simulating a constant evaporation rate of 2 mm/day for 100 days, the number of time-variable boundary records is 1, which indicates that 1 constant evaporation rate is being simulated. However, when using variable daily weather data, the number of time-variable boundary records is 100; you should later then provide a time series that comprises 100 evaporation rates, one for each day being simulated (there will be a table having 100 empty lines under “Variable Boundary Conditions”, which should be filled with the evaporation data).

Click next....

1.1.4 Print Information

Print Information

- Time Level Information: If this option is checked, then detailed results of fluxes, pressure heads, and other variables are printed at each time step. This may lead in some cases to enormous output files, especially if the simulation time is long or the solution is not converging (time steps can become very small and HYDRUS writes to output files at the end of each time step). In such cases, the information provided might be redundant and may unnecessarily increase the run time. Manipulating extremely large output data sets may become problematic, and hence it may be advantageous to turn this option off if there is no real need for the transient data; in that case the information will be provided at prescribed print times only, which could be increased to provide the necessary details.
- Screen Output: This option decides whether or not results are dynamically shown on the computer screen during a simulation. We recommend to always use this option, especially for new projects so as to monitor their progress. It is, however, recommended to uncheck this option for “inverse solution”.
- Print Fluxes: If this box is unchecked, nodal fluxes will not be printed, that is, in the “Graphical Display of Results”, velocities will not be available. In ‘Convert Output to ASCII’, the 4th option ‘Velocities’ will not be available (the ‘V.OUT’ file will be empty).
- Print times: These are prescribed times at which detailed run information is printed to the output files, such as fluxes, pressure heads, water contents, and concentrations. The data provided at print times could be of particular significance in many ways:



1. They provide a full suite of results at specified transient times. Information in the following output files is only provided at print times: 'h.out', 'th.out', 'v.out', 'concx.out', 'sorbx.out', 'temp.out', 'boundary.out', and 'balance.out'.
2. They are used for animating the simulation, that is, the more print times the smoother the animation.
3. If T-level information is turned off, you will get fluxes only at the print times.

Number of Print Times: Specify the number of print times (250 maximum in the Graphical Interface; for a higher number see Section 4.12).

Select Print Times: The default option will provide equally spaced print times, that is, run time divided by the number of print times. You can enter times manually provided they are listed sequentially and are less or equal to the total run time. It is useful to choose the default option and then to manually change some of the default times. The interval between print times cannot be less than the minimum allowed time step. An optimal distribution of the print times generally depends upon the type of problem. Figure 1.3 holds for a high-conductivity soil (loamy sand, $K_{sat}=350$ cm/day) that is freely draining for 10 days. Most of the drainage takes place during the first half day of the simulation. To get a good animation, and to obtain useful nodal fluxes during the early drainage period, you must have most print times during this period (uniformly distributed default print times will then be less useful). If you prefer a large number of print times (50 or 100), it would be very tedious to enter these manually. It is then a good idea to prepare them first in a spreadsheet in **one row** (make sure the number is identical to that specified under 'number of print times'), copy the row, then click on 'Select Print Times' in HYDRUS, select the first cell, and use "Ctrl_V" to paste the print times. We include the spreadsheet "**Print times.xls**", which has templates for 10 to 250 print times. You then only need to provide the total run time. The resulting print times vary according to a power function where the user specifies the power (positive or negative; 0 results in uniformly-spaced times). Copy the results and paste into HYDRUS.

⊗ Whenever you change run time (especially if you shorten it), don't forget to change print times

Free or Deep Drainage Boundary Flux

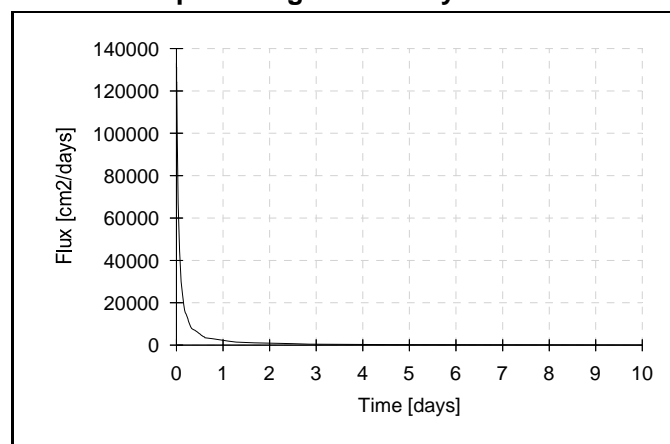
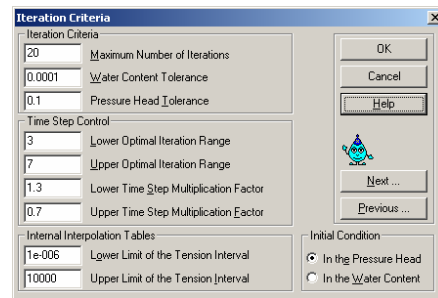


Figure 1.3: Example where closely spaced print times are required

1.1.5 Water Flow Iteration Criteria

Iteration Criteria, Time Step Control, and Internal Interpolation Tables are well described in the built-in online HELP in HYDRUS. Part of the following description about this window was extracted from the HYDRUS Help.

Because of the nonlinear nature of the Richards equation, an iterative process must be used at each new time step. This iterative process continues until a satisfactory degree of convergence is obtained, i.e., until the change in pressure head (or water content) at all nodes between two successive iterations becomes less than a small value (i.e., the absolute pressure head (or water content) tolerance).



- Iteration Criteria

Maximum Number of Iterations: The maximum number of iterations allowed during any time step. If the maximum number of iteration is reached without reaching a solution, the time step is divided by 3 and the computation at the current time level restarted.

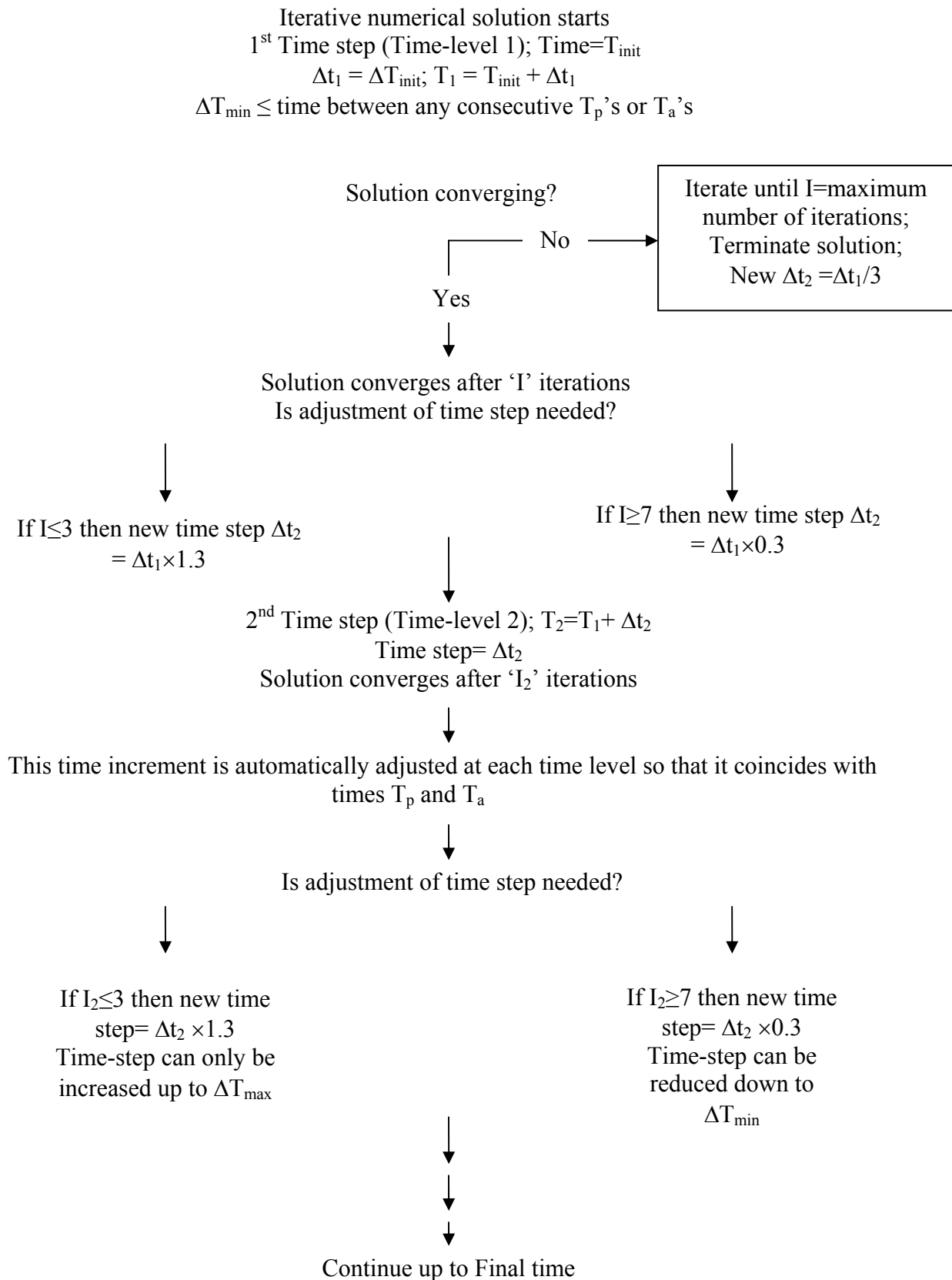
Water Content Tolerance: Absolute water content tolerance for nodes in the unsaturated part of the flow region. This parameter represents the maximum allowed absolute change in the value of the water content between two successive iterations during a particular time step.

Pressure Head Tolerance: Absolute pressure head tolerance for nodes in the saturated part of the flow region [L]. This parameter represents the maximum allowed absolute change in the value of the pressure head between two successive iterations during a particular time step.

- Time step control

Refer to online Help for details about the optimal iteration range and the time step multiplication factor. The information required in this window relates mainly to the numerical time-marching scheme. We recommend not to change the default values unless you are an advanced user. For more details on ‘Time Control’, refer to the HYDRUS Technical Manual (p.48). Chart 1.1 demonstrates the time marching scheme and explains the meaning of the various parameters in this window. Note that for Chart 1.1:

- o The indicated ‘Time Step Control’ parameters are all default values.
- o ΔT_{\min} and ΔT_{\max} are the minimum and maximum allowed time steps, respectively (specified in the window “Time Information”), ΔT_{init} is the initial time step, and T_{init} the initial time.
- o Time discretization associated with implementing the boundary condition (herein called T_a): refers to times when the imposed BCs change (such as variable pressure heads or flux BCs), or times when the introduced fluxes change (such as precipitation in atmospheric BCs).
- o The time discretisation associated with print times is T_p .

**Chart 1.1:** Time marching scheme in HYDRUS

- 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. Doing so has been shown to speed up the calculations.

Interpolation in the tables can be avoided by specifying both values equal to zero. Then the soil hydraulic properties are always evaluated analytically directly from the hydraulic functions, i.e., without interpolation. The soil hydraulic properties are also evaluated directly for values of the pressure head outside of the pressure head interval.

Lower limit of the tension interval: Lower limit [L] of the tension (absolute value 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 limit [L] of the tension interval for which a table of hydraulic properties will be generated internally for each material.

- Initial Conditions

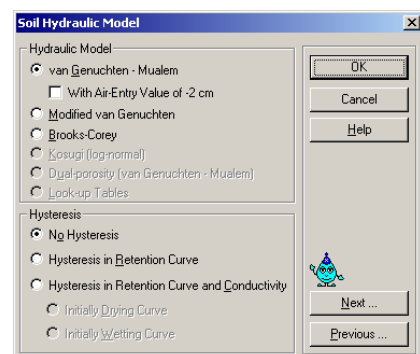
Initial water flow conditions can either be described in terms of volumetric water contents or pressure heads; they describe the state of the system prior to the simulation. The initial conditions themselves are later set in “Boundary Conditions Editor/Initial Conditions”. There are two options, pressure-head or water content. It is the “Initial Condition” option here under “Iteration Criteria” that will decide whether the initial soil conditions to be entered later are to be interpreted as water contents or pressure heads.

The ‘Iteration Criteria’ and ‘Internal Interpolation Tables’ parameters may be changed in cases where numerical instabilities occur because of extreme conditions; see Section 6.4.

1.1.6 Water Flow-Soil Hydraulic Model

HYDRUS allows users to select three types of models to describe the soil hydraulic properties: van Genuchten (1980), Brooks and Corey (1964) and modified van Genuchten type equations (Vogel and Cislérova, 1988).

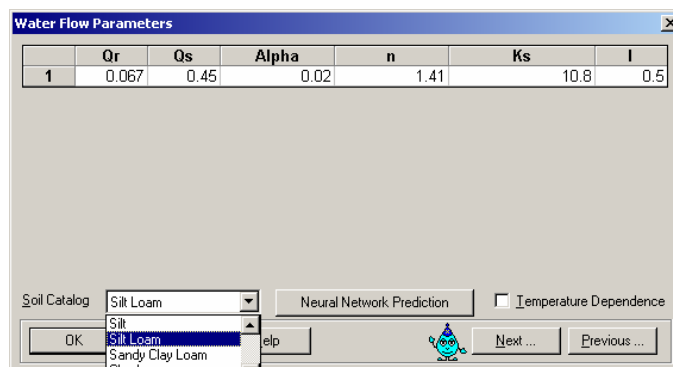
Those models describe the water retention parameters of the soil as well as the hydraulic conductivity function, often referred to also as the constitutive relationships. They relate water content and hydraulic conductivity to the pressure head. A detailed explanation of these models is found in Appendix I. For details on hysteresis, refer to Section 4.4.



1.1.7 Water Flow-Soil Hydraulic Parameters

The required input parameters vary with each model. The meaning of the input parameters for each of the three models is outlined in detail in Appendix I.

More parameters will be required if hysteresis is modelled, see Section 4.4. If you have directly measured estimates, these can be entered into the “Water Flow Parameters” window.

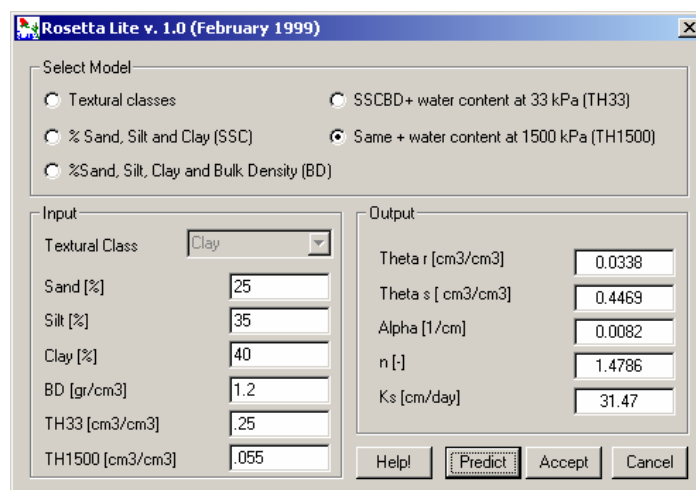


HYDRUS provides two levels of help in providing users with soil hydraulic parameters. In cases where only the soil textural class is known (e.g., silt or clay loam), a drop down list (shown above) provides all of the required parameters. The user must accept the fact that these only represent very approximate averages for the different textural classes. The parameters were taken from Carsel and Parrish (1988). Another possibility is to use the Neural Network Prediction option.

Neural Network Prediction (Rosetta)

HYDRUS was coupled with the Rosetta Lite DLL (Dynamically Linked Library), which was independently developed by Marcel Schaap at the U.S. Salinity Laboratory. Rosetta implements pedotransfer functions (PTFs) which predict van Genuchten (1980) water retention parameters and the saturated hydraulic conductivity (K_s) in a hierarchical manner using soil textural class information, 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.

Click the “Neural Network Prediction” button and this window will appear.



Depending upon the availability of soil information, you have five options; select the appropriate option under “Select Model”. The relevant cells will activate accordingly:

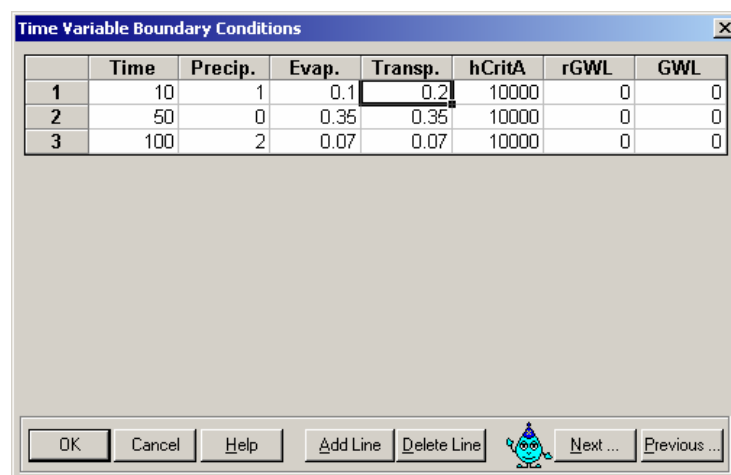
1. Textural class: you are prompted to only enter the textural class, more or less like the built-in drop-down list in HYDRUS.
2. % Sand, Silt, and Clay; enter the required percentage values (SSC).
3. As 2 above (SSC), but additionally enter the measured bulk density (BD) (g/cm^3).
4. As 3 above (SSCBD), but also enter the measured water content at 33 kPa (3.3 m, or one third bar).
5. As 4 above, plus the measured water content at 1,500 kPa (150 m, or 15 bar).

Once you finish, click “Predict”. If you are not happy with the result, re-enter values and re-predict. Finally, click “Accept”, which will automatically import the predicted values into the “Water Flow Parameters” window.

1.1.8 Variable Boundary Conditions

In this window, the user is prompted to enter boundary conditions that vary with simulation time. These conditions are dynamic (variable) through the simulation but static (constant) through a defined period of time. That is, the modeller discretizes the total simulation time into portions with different boundary values. For example, refer to the precipitation values below:

0-10 days 1 cm/day
 10-50 days 0
 50-100 days 2 cm/day



The number of rows (3 here) in this window depends on the number of “Time Variable Boundary Records” specified earlier in the “Time Information” window. Note that the last “Time” must be equal to the “Final Time” specified in the “Time Information” window.

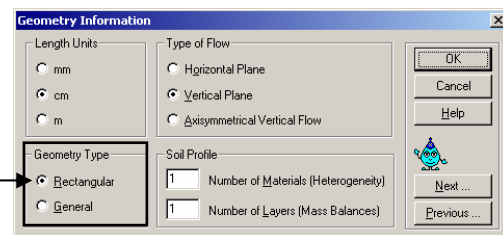
Time	Time for which a data record is provided (T)
Precip	Precipitation or rainfall rate (L/T)
Evap	Potential evaporation rate (L/T)
Trans	Potential transpiration rate (L/T)
hCritA	Absolute value of the minimum allowed suction at the soil surface (L). Refer to Appendix II for more details.
rGWL	Drainage flux (L/T) across the bottom boundary (inflow is negative), or other time-dependent prescribed flux boundary condition; set to zero when no time-dependent flux boundary condition is specified.
GWL	Groundwater level (L), or other time-dependent prescribed head boundary condition; set equal to zero when no time-dependent head boundary condition is specified

More details are found in Section 1.1.10 under points 4, 5, and 9.


1.1.9 GEOMETRY AND FINITE ELEMENT MESH EDITOR

1.1.9.1 Rectangular Grids

When the basic shape of the domain is rectangular, this is the quickest way to prepare the finite element mesh. Actually, it is the only option in the absence of the MeshGen2D grid generator. The use of Rectangular grids makes it possible to construct an essentially rectangular domain made up of a number of triangular elements. The domain is a simple continuous square or rectangle with some undulating surface topography.

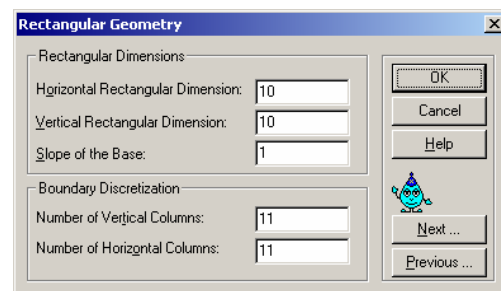


This option is activated when you select the “Rectangular” option of geometry type. Please go back and do so as needed, and then click OK.

Next double click ‘Geometry and Finite Element (FEM) Editor’ (or ); this window will appear. Enter values as shown.


Horizontal and Vertical Dimensions: Enter the dimensions of the domain.

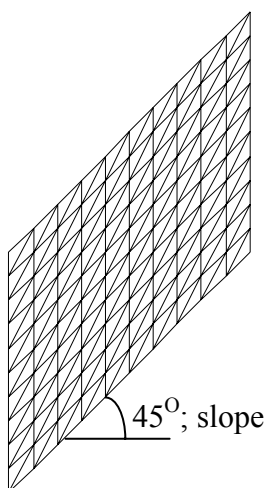
Slope of the Base: refers to the slope of the entire mesh.



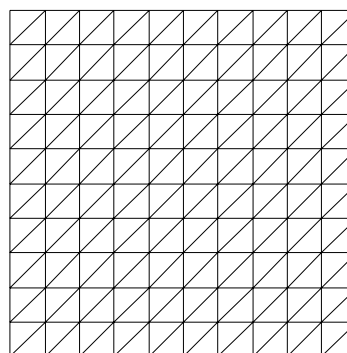
Number of Vertical and Horizontal Columns: refers to the number of rows and columns that will be used to construct the mesh; the minimum number is two. Two columns are needed to simulate one dimensional problems (see Test1, 2, 4, 5, and 6 in the Workspace 'Direct'). The maximum allowed is 10,000.

Click next....

If you accept all default options in the next window, you'll get a grid with a slope of 45° , (slope=1, which is the tangent of 45° ; see Figure 1.4 left side). However, if a slope=0 is selected, you will get the grid shown in Figure 1.4, right side. You can view these meshes graphically in the "Boundary Conditions Editor ()".



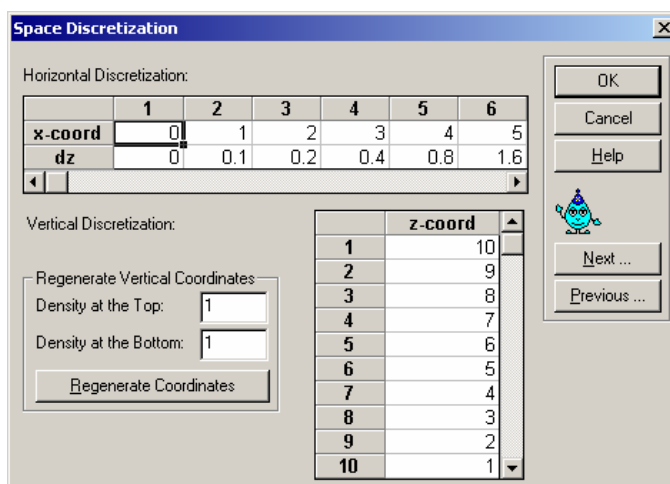
Slope of base = 1



Slope of base = 0

Figure 1.4: Rectangular grids with different slopes

Horizontal Discretization: The default discretization in the horizontal (x) direction is simply the horizontal dimension divided into the number of vertical columns minus one. You can overwrite these values by entering new values manually into the individual cells. You can also prepare the coordinate data set in a spreadsheet, and copy it; click the relevant cell and then Ctrl_V to paste the data into HYDRUS.



You have the option of introducing an undulating surface topography as shown here. This is done manually; click each cell along “dz” and enter a vertical increment (positive or negative) to the surface node coordinate. The resulting mesh is shown in Figure 1.5.

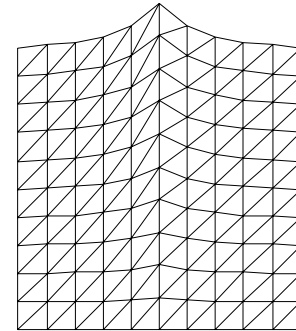


Figure 1.5: Rectangular grid with surface topography

Vertical Discretization: You have more control over the vertical discretization. You can change the density of the vertical elements either from the top or from the bottom.

Density at the Top, and Density at the Bottom: The densities decide the mesh sizes; the smaller the density the finer the mesh near that end. The only thing that matters is the ratio between the two values. Assigning any set of equal values for both the top and the bottom will result in the default uniform vertical discretization, that is, elements along that column are of equal height.

To obtain a finer mesh at the top end of the grid, specify for example 0.1 (Top density/Bottom density < 1) for “Density at the Top” then click Regenerate Coordinates. Notice that the distances between coordinates in the vertical column are now smaller near the top than near the bottom. Again, you can view the mesh in the “Boundary Conditions Editor (🗨)”.

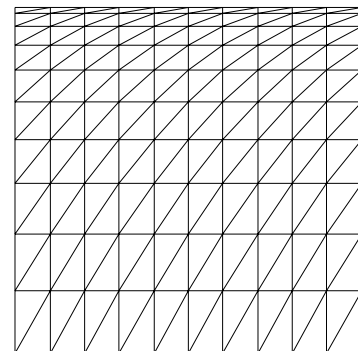
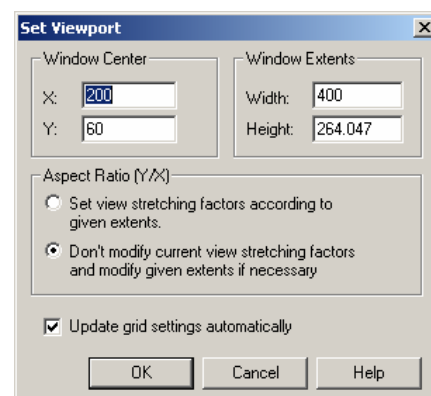


Figure 1.6: Rectangular grid with fine density at top

1.1.9.2 General Grids; Meshgen in HYDRUS-2D

If under ‘Geometry Information/Geometry Type’, you chose ‘General’, HYDRUS will proceed as follows to create your mesh using the embedded finite element mesh generator ‘MeshGen2D’. Go first to “Geometry and FEM Generator” (or 🗨).

The **Set Viewport** window (may be edited later, see [View/Viewport](#)) will appear.



- **Window Centre and Extents:** You should know the physical size of your problem, that is, the width and height of the domain so that the screen can accommodate and make it visible. Enter the window centre and

extents using the units (e.g., cm) that you specified earlier (in ‘Geometry Information’).

- **Aspect Ratio:** The aspect ratio (Y/X dimensions) of your domain does not necessarily match that of the screen; you have two options:

Either keep the extents and stretch (deform) the domain, or keep the stretching factor (default value=1 keeps original shape of grid) and modify the domain extents (HYDRUS will accommodate the larger of the two dimensions and change the other one according to the aspect ratio (about 0.67)).

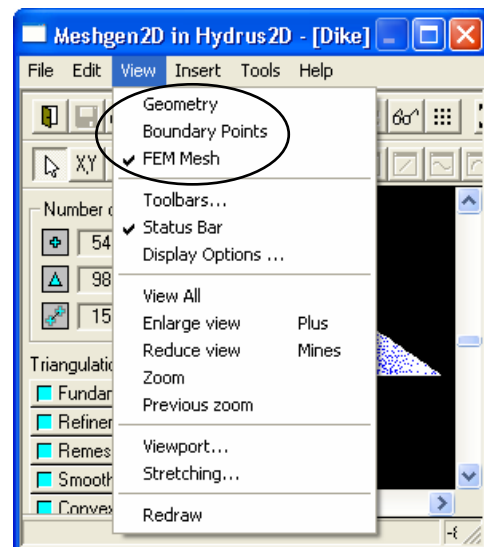
The latter option is the default and is the preferred option; it is generally better to start with an undeformed domain to get a good feeling of the problem; you can always change the stretching factor later.

Upgrade Grid Settings: It is also useful to have Grid Setting change automatically according to the size of the domain so leave the box checked. Alternatively, the Grid Settings may be changed manually using Tool/Grid Settings.

There are basically three stages during construction of the finite element mesh; they are found under ‘View’ or at the toolbar and should be processed sequentially:

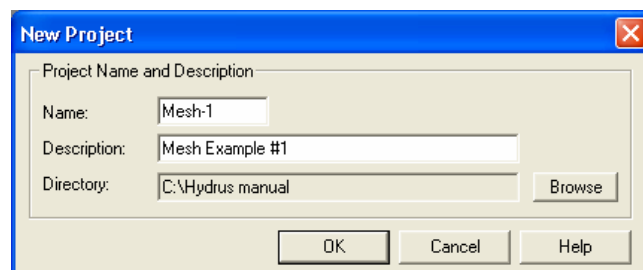
- 1- **Geometry** (📐) lets you draw the domain of the problem,
- 2- **Boundary Points** (📍) lets you control the density of the mesh by controlling the density of boundary points (external or internal), and
- 3- **FEM Mesh** (🔗) actually creates the mesh. If you are not happy with the results then repeat the three steps again.

We will now go through a few mesh examples that cover the most important aspects of the Meshgen package.



Example 1: Draining a soil block next to a stream; Project ‘Grid-1’ (CD\HYDRUS Manual Projects\Sect-1)

Start a new project as shown; select a suitable directory using the Browse button.



We will create a mesh that represents a soil block located next to a shallow stream and drained along the other side. The block is 6 m wide and 2 m high.

The following example demonstrates how the finite element mesh is constructed. To do so we will skip all other options and start directly with the Mesh Editor. We will use the default length units (cm).


Double click on Geometry FEM Mesh Editor (or ).


Window Centre: 300,200

Window Extents: 650,400 (it is a good idea to make the extents slightly larger than the maximum dimension in the grid to ensure that the 0,0 origin point is visible). If the origin is not visible, use the scroll bars to show it.

Leave the remaining options at default.

We will draw part of the grid graphically (using the mouse). With this option, coordinates can be precisely located only using the “snap to grid” option, which is turned on by default. You can change the grid spacing to suit your needs.

Tools/Grid Settings (or ); change the grid spacing to 10,10. When the grid spacing is too fine compared to the dimensions of the view window, the grid will not be displayed, even if active in the background.

Insert Line/Graphically (or ); a special cursor (a pair of cross-hairs) will appear. Click on the following coordinates (coordinates appear dynamically on the lower RHS of the screen):

(0,0), (600,0), (600,200), (90,200)

Right-click to end.

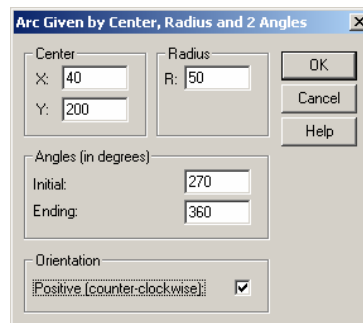
We have drawn a polyline consisting of three segments (still one entity)

Tools/Arc Input Mode: Select Centre, Radius, and Two Angles (or )

Insert Arc: Select Numerically (or use )

Enter the data as shown here and click OK.

We will now close the grid by drawing two lines numerically.



Insert Line/Numerically (or use )

Under Current Point

X:40, Y:150


click Add New Point.

X:0, Y:150

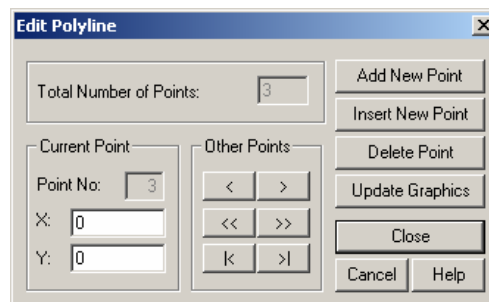
click Add New Point.

X:0, Y:0

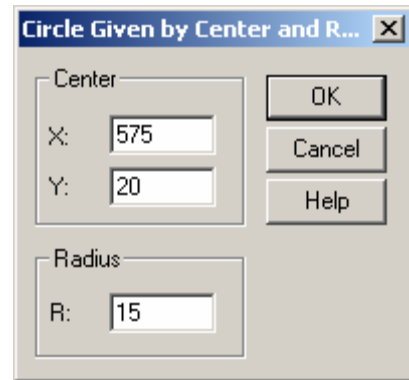
click Close.

Tools/Circle Input Mode: Select Centre and Radius (or )

Insert/Circle: Select numerically ()



Enter coordinates as shown here.



View/Boundary Points (📍), Discretization of Boundary Nodes

From the sidebar select: Number of Points

The cursor will turn into a hand. Click any boundary to find the number of points on that boundary. In cases where you have only one boundary, the number defaults to 120. The default number of points in this case is

Nodes on outer boundary =113

Nodes on inner circle =15

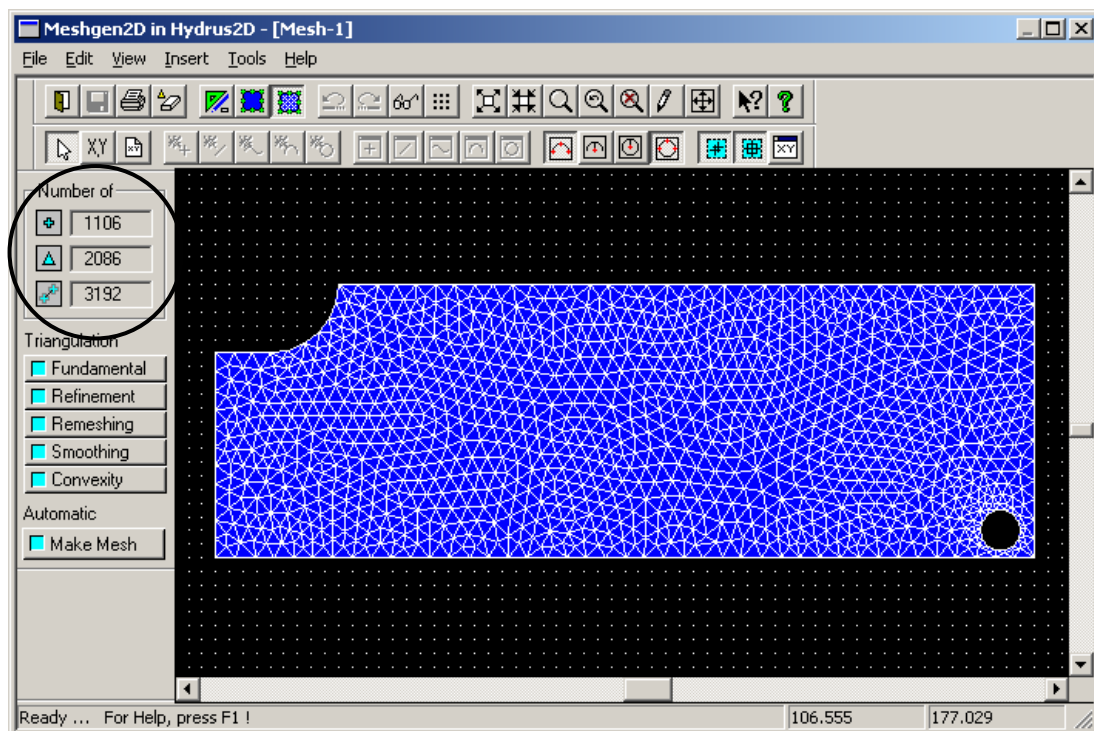
Note that we included this step only for demonstration purposes.

View/FEM Mesh (📊), Finite element discretization

From the sidebar select: Make Mesh

You should see the mesh shown below.


File/Close and Return (🔼) takes you back to HYDRUS,
Save changes? YES



Referring to the MeshGen window on the previous page, the encircled numbers refer to the following:

- 1106 Number of nodes.
- 2086 Number of mesh-triangles (elements).
- 3192 Number of edges.

Alter the density of the mesh: produce a coarse uniform mesh

Double click on Geometry and FEM Mesh Editor (or ):

View/Boundary Points

Number of Point (sidebar); a warning will appear, click YES.

Click anywhere on the outer boundary and enter 80.

View/FEM Mesh ()

Make Mesh (sidebar)

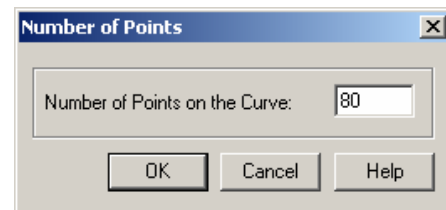


Figure 1.7 shows that the resulting mesh is much coarser.

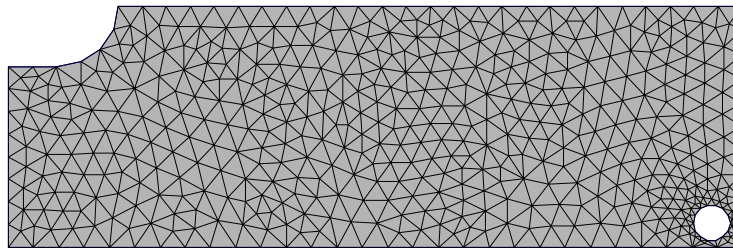




Figure 1.7: Finite element mesh with fewer boundary points

File/Close and Return ()

Save changes? YES

Alter the density of the mesh: produce a mesh of varying density

Double click on Geometry and FEM Mesh Editor (or ):

View/Boundary Points ()

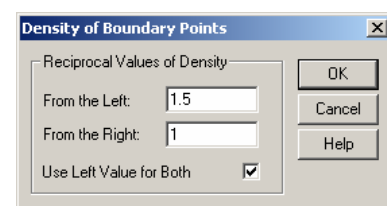
All default (sidebar); this will bring back the default values (outer number of nodes=113)

We will now try to alter the density of the node distribution along the outer boundary to get a coarse mesh on the lower LHS and a finer mesh on the lower RHS.

Fixed Point/Density (sidebar)

Select the lower LHS fixed point; pink coloured point will turn yellow.

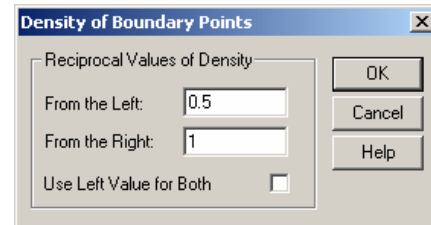
Right click and enter the values as shown.



Note that if you want the density to be equal on both sides, leave the box “Use Left Value for Both” checked and enter a value in “From the Left”.

The arrows in Figure 1.8 show the convention used for right and left (notice that you are always looking at the grid from the outside).

Select the lower RHS fixed point, and enter the values shown here. Note that when value is <1 this results in a finer mesh and vice versa.




View/FEM Mesh (),
Make Mesh (sidebar)

Figure 1.8 shows that the lower boundary has an increasing number of elements towards the RHS of the mesh.

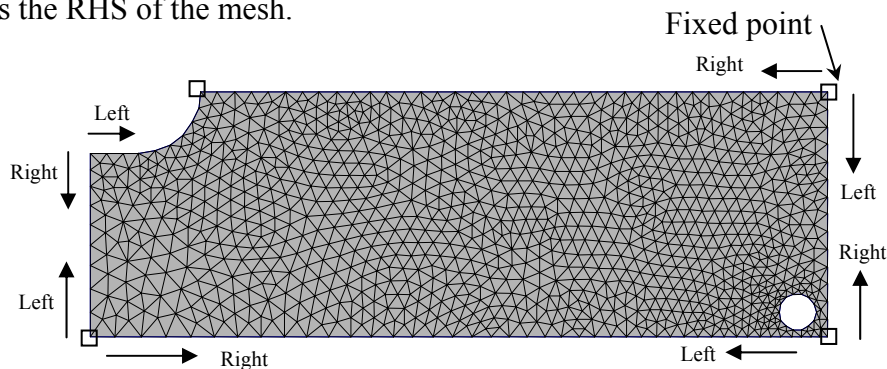





Figure 1.8: Convention for changing mesh density

File/Close and Return (),
Save Changes? YES

Example 2: Hill, 30 m wide sloping at 15%; Project ‘Grid-2’ (CD\HYDRUS Manual Projects\Sect-1)

Start New Project:
Project Name: Grid-2

Double click on Geometry and FEM Mesh Editor (or ):
Enter the following extents: Centre; 1500, 500 Extents; 3500, 1000

Insert Line/Graphically (or ); click on the following coordinates:

(0,0) (3000,0) (3000,500) (0,500) (0,0) Right-click to end

We will now introduce a slope to the square grid.

Tools/Manipulation Mode/Numerical

From the side menu: click Move

Select the upper LHS node; enter the new Y-coordinate=950 ($500+3,000\times 15\%$) OK

Select the lower LHS node; enter the new Y-coordinate=450 ($0.0+3,000\times 15\%$) OK

We have a neutron probe at a distance of 1000 cm from the LHS of the mesh measuring water contents at four depths (100, 200, 300, and 400 cm from the soil surface). To be able to compare simulated and measured values, we must make sure that nodes exist at these locations.

Insert Line/Numerically (or)

Under Current Point

X:1000, Y:700 (at x=1000, soil surface level is 800 cm) click Add New Point

X:1000, Y:400 click Close

View Boundary Points (or)

Fixed Points/Insert (sidebar, inserts one point at a time)

Click on 1000,600 and 1000,500 (it is best to do this with the snap option on).

This should result in a line having four fixed points at the desired locations (by adding fixed points you are making sure that nodes exist at these locations).

You can control the density of the mesh by changing the number of points on the line, which defaulted to 15. We will slightly coarsen the mesh around the line:

Boundary/Number of Points (sidebar)

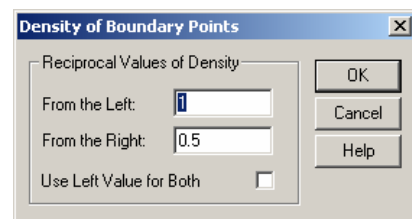
Click on the line and change to 10

We now refine the upper part of the mesh (only the area above the inserted line). To do this we need to add a fixed point to define a density around it:

Fixed Points/Insert (sidebar)

Click on 1000, 800

Fixed Points/Density (sidebar); select fixed point (1000, 800), right-click and enter values as shown here.



Fixed Points/Density (sidebar), select fixed point (0, 950), right-click, and enter the following values:


From the left: 0.5

From the right: 1

Uncheck 'Use Left Value for Both'.

View/FEM Mesh (or)

Make Mesh (sidebar) (final result shown in Figure 1.9)

File/Close and Return ()
Save Changes? YES

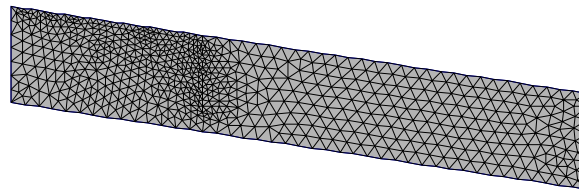



Figure 1.9: Sloping mesh with high density along a vertical axis

Example 3: Hypothetical mesh; Project ‘Grid-3’ (CD\HYDRUS Manual Projects\Sect-1)

Start New Project:

Project Name: Grid-3

Double click on Geometry and FEM Mesh Editor (or ):

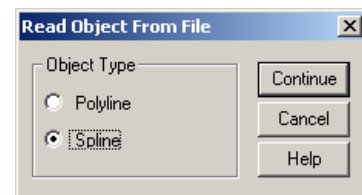
Enter the following extents: Centre; 8, 0 Extents; 25, 17

Insert/Object from File

Select Spline

Enter file name: Wavy Sine Mesh.txt (CD\HYDRUS Manual Projects\Sect-1). Click Help to read more about file formatting.

Continue: change file type to ‘all files’ since default is ‘.h2d’.



You should see a boundary as shown in Figure 1.10. The coordinates were generated in a spreadsheet using a sine function. This option is useful when you have a large surveyed landscape (e.g., survey data for a large hill slope).



Figure 1.10: Imported spline from file

Use the scroll bar at the bottom of the screen to move the view to the right until the point (0,0) is near the RHS of the screen.

View/Enlarge View (🔍) 2-3 times to zoom in on point 0,0 and leave some working space to the left.

Tools/Grid Settings/Snap off (🛑) (do not use the snap option since the coordinates of the spline do not conform to the grid)

Insert/Spline/Graphically (👉) (we could draw this exactly using the numerical option, but in this exercise we try to demonstrate most of the available features)

1st point: (0,0) (end of sine curve)
 2nd point: (-0.5, -0.5) (approximately)
 3rd point: (-1.5, -0.9) (approximately)
 4th point: (-3.0, -1.0) (approximately) Right-click to end.

Tools/Manipulation Mode/Numerical

Object's Points/Move (sidebar)

Select the last point on the spline, and change the coordinates to exactly -3.0, -1.0 (with snap off, you can never get precise coordinates graphically).

View/View All (🔍)

Tools/Grid Settings/Snap on (🛑)

Tools/Manipulation Mode/Graphical

Select the wavy boundary by clicking on the line between the nodes (should turn yellow when selected).

Press and hold the SHIFT key; select the other spline also.

From the Sidebar

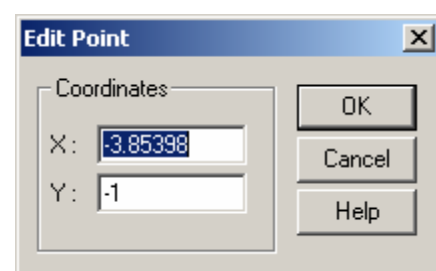
Selection Move (cursor turns into a square)

Select the middle peak on the sine-shaped boundary; the entire drawing should now move when you move the cursor. Click on nearby grid point (7,1) (we want the peak to coincide with a grid point so that we can draw a horizontal line that starts from a point directly underneath the peak).

Now we want to know the coordinates of the end of the spline.

Object's Point/Move

If you click on the last point of the spline, you'll see its coordinates as in this figure. The x-coordinate (unknown because we moved the grid) will be highlighted; right click on this coordinate and copy it to the clipboard. Select Cancel.



Alternatively,

View/Zoom (🔍)

Select a small window around the end point of the spline (high magnification)

Edit/Geometry/Object's Points/Coordinates

Copy the x-coordinate onto Clipboard

View/Zoom Previous (🔍)

Insert/ Line/Numerically (draw Polyline)

(7, -5)

Add new point

(-3.85398, -5) (you can paste the x-coordinate from the clipboard) Add new point

(-3.85398, -1)

Close

Tools/Manipulation Mode/Graphical (the snap feature should be on)

Select the Polyline and hold the SHIFT key to select the spline also.

Selection/Reflect: you'll be prompted for an axis for reflection,

Click points 7,1 (peak of sine shape) and 7, -5 (end of Polyline).

The boundary should now look as in Figure 1.11.

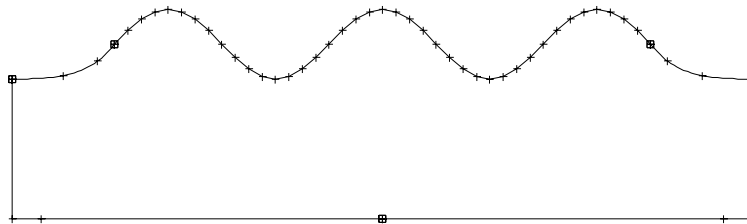


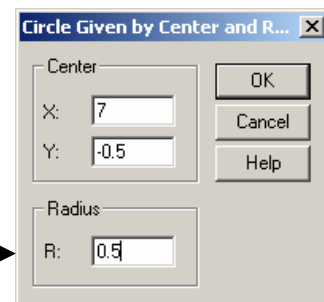
Figure 1.11: Outer boundary of domain

We now vary the density of the grid around the middle mound.

Tools/Circle Input Mode/Radius and Centre (or 📍)

Insert/Circle/Numerically (or 📍)

Enter the coordinates as shown here.

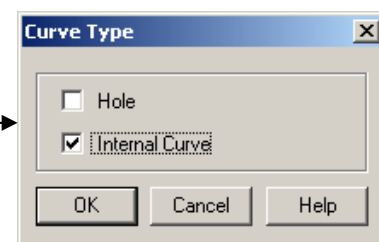



View/Boundary Points (or 🗺️)

Hole

Select the circle

Check 'Internal Curve' as shown here. This will convert the circle from an opening in the transport domain to an internal curve that will help us change the local grid density.



View/Boundary Points (or )

Number of Points: click on the circle and change the number to 25.

View/FEM Mesh (or )

Make Mesh (sidebar) (final result shown in Figure 1.12)

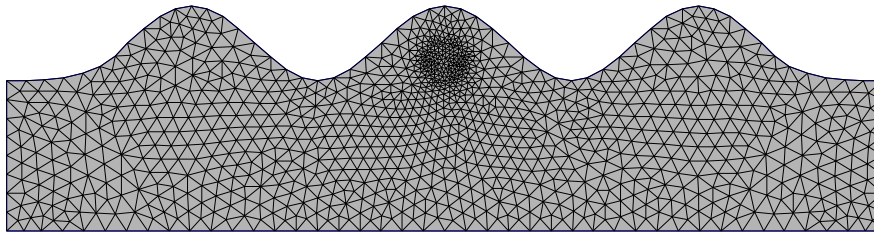



Figure 1.12: Final mesh

File/Close and Return (or ),

Save Changes? YES

Several other useful features exist that have not been covered:

FILE, you can:



Print the mesh ()

Print Preview before printing

Copy mesh to clipboard and paste into another software


Save the mesh details into a text file

Edit, you can:

Undo/redo (/); is very useful to reverse the last action


Geometry split/merge; lets you join and split similar objects like polylines or splines.

View/Display Options; you can show/hide edges, nodes, elements, and their numbers.

Tools/Check Geometry (); this is usually done automatically when you finalise the geometry of the mesh and move to the boundary points. You can also check the integrity of the grid manually using this option (e.g., the outer boundary must form one continuous closed shape, and/or no intersecting lines can exist in the grid).

Tools/Find; you can find a particular Node/Edge/Triangle by entering its number.

Tools/Point Picking; when checked during drawing in Graphical Mode, a square appears around the cross hairs of the cursor. This causes one to snap to an existing point (when the square encloses the point). This snap-to-point feature can be very useful; we recommend leaving it on.

Help/Context Sensitive (); click this option and then click on any command in both the drop-down and sidebar menus and you will get help on them.

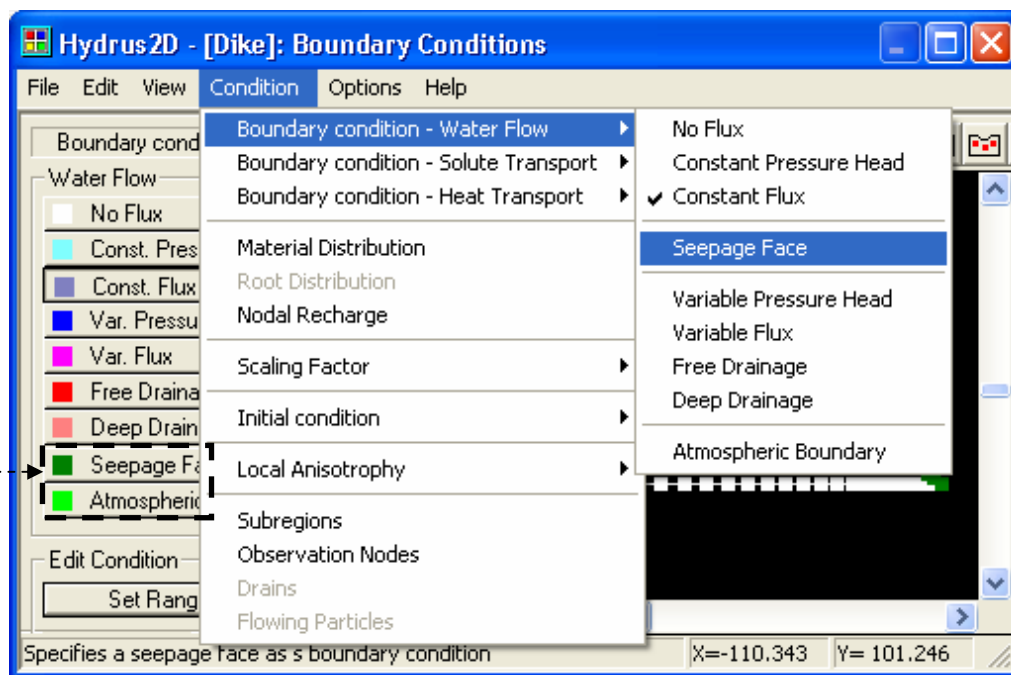
Note that “**Insert/Periodical Condition**” is outside the scope of HYDRUS.

1.1.10 Boundary Conditions and Domain Properties Editor

Specifying appropriate boundary conditions (BCs) is one of the most critical tasks when constructing a numerical model. A water flow BC is a known value of the flux, head, or gradient along the outer boundary of the FE mesh (it means the external boundary of the selected flow domain; it is the interface between the soil and the outside world). Solving the governing equations for saturated/unsaturated flow, which means finding the new head at each node in the FE mesh in a time-marching scheme, requires knowledge of those BCs. Otherwise the problem becomes mathematically indeterminate. The BCs in a model must simulate real-life conditions and hence must be selected with extreme care.

In HYDRUS, boundary conditions are categorised as follows:

- **System-dependent:** or dynamic BCs, meaning that they can change during the simulation (i.e., they depend on the solution at the end of each time step). They may depend on saturation conditions (as in a seepage face or a drain), or on soil properties and/or climate conditions (as at soil/atmosphere interfaces).
- **System-independent:** This type of BC is entirely known a priori, is implemented by the user, and is independent of the simulation results.




System-independent water flow BCs include:

- A known head (as in Constant Pressure and Variable Pressure)
- A known flux (as in No Flux, Constant Flux, Variable Flux, and Deep Drainage)
- A known gradient (as in Free Drainage)

Refer to the HYDRUS Technical Manual p. 24-26 for more details.

1.1.10.1 Water Flow Boundary Condition (🌊):

A Water flow BC is set as follows:

- Double-click on **Boundary Condition Editor** (or )
- The appropriate condition is selected from the drop-down menu (Condition/Boundary Condition-Water flow) or from the buttons located on the left hand side of the window (this option is quicker). The cursor turns into a finger-like pointer.
- Left-click and drag on the required nodes, and left-click to end the node selection, or right-click to abort this task.
- If a certain value is associated with the BC as in ‘Constant Head’, ‘Constant Flux’, or ‘Deep Drainage’, a window will appear where you are required to enter the relevant data.
- Repeat the procedure in case of multiple BCs.
- If you have made a mistake, you can make changes using the LHS buttons while the appropriate BC button is still activated. Use “Edit condition” and then “Select”, which activates “Set value” that changes the boundary value.

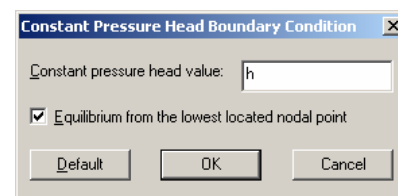
We will now explain in detail the meaning of each water flow BC:

1. **No Flux:** this BC means that the boundary is sealed from the outside world; no water flows into or out of the domain through this boundary; all boundary conditions default to this option if nothing is specified.
2. **Constant Head:** this BC refers to a constant pressure head during the entire simulation. Values for the pressure head (positive or negative) are entered in the window shown below. We recommend that the initial conditions be set first followed by this BC (if the reverse is done, initial conditions will overwrite BCs, since constant head BCs are stored in the same vector as the initial condition). We illustrate a variety of cases where a constant head boundary condition is applicable (refer to Figure 1.13):

BC-1: A tension disc infiltrometer; head = $h_1 - h_2$ (negative) on the nodes representing the soil immediately beneath the disc.

BC-2 and BC-3: A constant head from an overflowing tank or a Mariotte Bottle assembly; head = h .

BC-4: Linearly varying hydrostatic head on nodes representing the soil beside a stream. When prompted for the head value, check the “Equilibrium from the lowest located nodal point” box and enter the value for head at the lowest point, which is equal to ‘ h ’.



BC-5: A constant negative head resulting from a hanging water column assembly; head = $-h$.

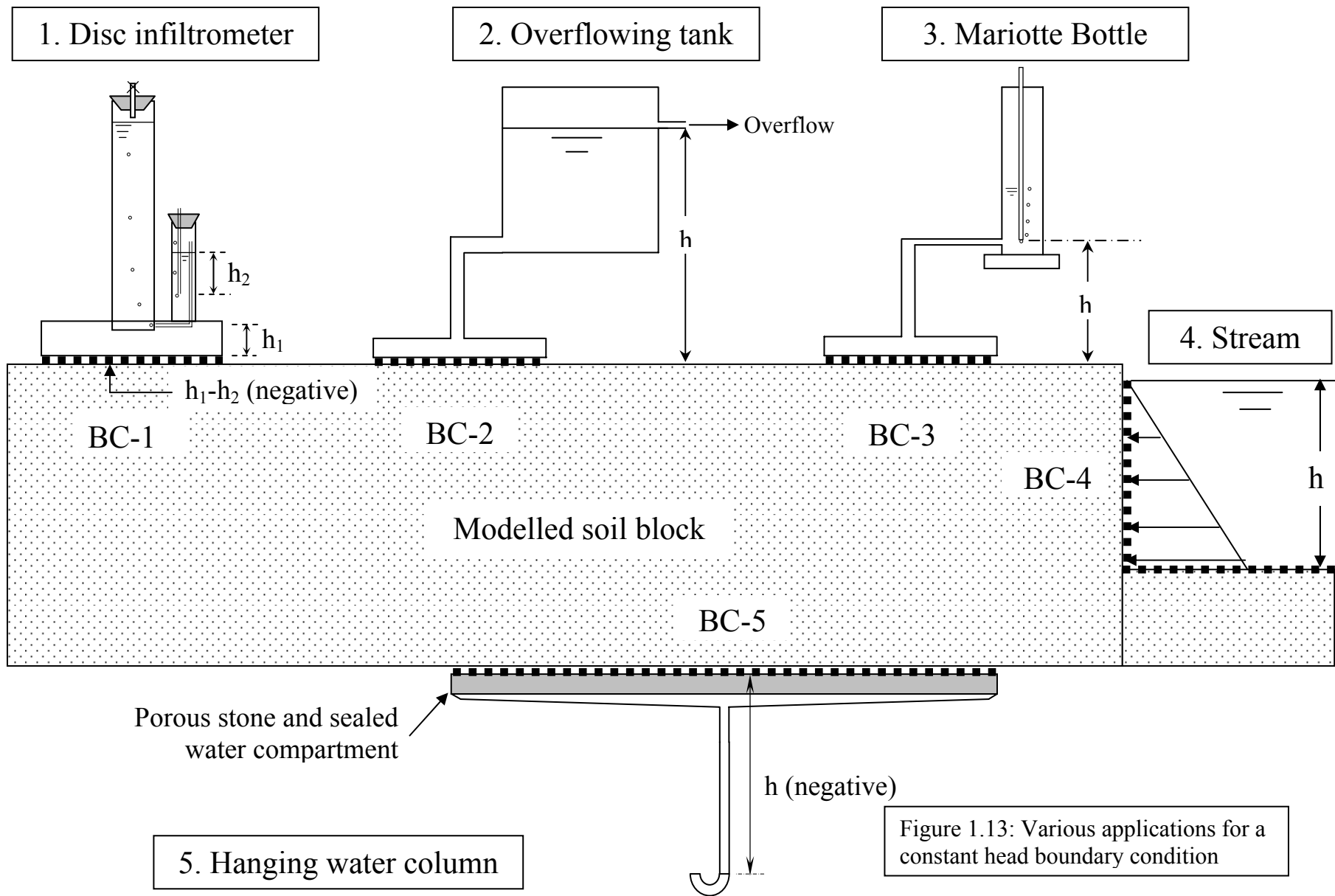


Figure 1.13: Various applications for a constant head boundary condition

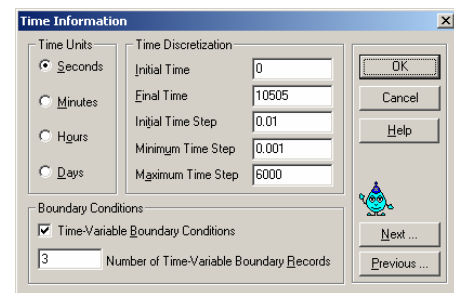
3. Constant Flux: this BC refers to a constant flux during the entire simulation. The flux (v) has the units (Length/time, L/T), like rainfall rate (cm/day). The flux is applied over a certain width of the outer boundary of the FE mesh (W). The applied flux is entered after selection of the nodes.

If you need to know the total flow rate over the constant-flux boundary (Q , L^2/T), the flux is integrated (summed) over the width (W); $Q (L^2/T) = v (L/T) \times W(L)$. Refer to p. 2.9 for an example.

4. Variable Pressure: this BC refers to a variable pressure head that is input as a time series. The time series has as many records as the number of times the head changes over the boundary (you can enter as many records as you like). For each record in the time series, you enter a constant head that is effective during the period between the time of the previous record and the current one.

This BC may be applied also to a sloping or vertical surface where the specified head is assigned to the lowest vertical coordinate of the boundary; the upper points are then assigned heads that satisfy hydrostatic equilibrium (e.g., a variable water level along a stream bank; see Section 4.1). Another example application is on a horizontal surface as in a multi-stage, tension disc infiltrometer experiment. We refer to one of the examples included in the HYDRUS package: Project 'Disc', which is found in the 'Inverse examples' workspace. That example considers a three-stage tension disc infiltrometer experiment.

- Check 'Time Information/Boundary Conditions', and enter 3 in the box (as shown).



- Under 'Variable Boundary Conditions', enter the Time and GWL (applied head). Referring to the figure (h_{CritA} is irrelevant here):
 - $0 < \text{Time} \leq 3400$ seconds, $h = -20$ cm
 - $3400 < \text{Time} \leq 6450$ seconds, $h = -10$ cm
 - $6450 < \text{Time} \leq 10505$ seconds, $h = -3$ cm

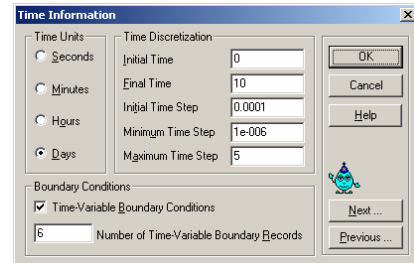
	Time	Precip.	Evap.	Transp.	hCritA	rGWL	GWL
1	3400	0	0	0	100000	0	-20
2	6450	0	0	0	100000	0	-10
3	10505	0	0	0	100000	0	-3

Prescribed
Time-variable
BC

5. Variable Flux: this BC refers to a variable flux that is input as a time series. The data is input in a similar manner as the ‘Variable Head’ boundary, but instead of ‘head’ (GWL) enter ‘flux’ (rGWL) in the time series. An example is drip irrigation with variable intensity.

We refer to one of the examples included in the HYDRUS package: Project ‘Drip’, which is found in the ‘Direct1’ workspace. In that example, the intensity is varied 6 times during a 10-day period.

Check the ‘Time Information/Boundary Conditions’, and enter 6 in the box (as shown).



- Under ‘Variable Boundary Conditions’, enter the Time and rGWL (flux). Referring to the window below:
 - i. $0 < \text{Time} \leq 1$ day, flux = -0.15 m/day (negative is inflow)
 - ii. $1 < \text{Time} \leq 2$ day, flux = 0
 - iii. $2 < \text{Time} \leq 3$ day, flux = -0.25 m/day
 - iv. $3 < \text{Time} \leq 4$ day, flux = 0
 - v. $4 < \text{Time} \leq 5$ day, flux = -0.25 m/day
 - vi. $5 < \text{Time} \leq 10$ day, flux = 0

	Time	Precip.	Evap.	Transp.	hCritA	rGWL	GWL
1	1	0	0	0.005	1000	-0.15	0
2	2	0	0	0.005	1000	0	0
3	3	0	0	0.005	1000	-0.25	0
4	4	0	0	0.005	1000	0	0
5	5	0	0	0.005	1000	-0.25	0
6	10	0	0	0.005	1000	0	0

6. Free Drainage: this boundary conditions specifies a unit gradient along the lower boundary (outflow, drainage) of the FE mesh. It is applicable in cases where the water table is located far below the domain of interest. This BC assumes a unit total vertical hydraulic gradient, that is, gravity flow with no pressure head gradient. This boundary condition should never be used along sides of the transport domain. It should be used only at the bottom of the domain.

7. Deep Drainage: this is a variable flux BC based on an observed experimental relationship between head and discharge. It is based on an expression proposed by Ernst and Feddes (1979) (as reported by Hopmans and Stricker, 1989):

$$q = ae^{b|h-h_0|} \quad (1.1)$$

where q is the discharge, a and b are fitting parameters (obtained from experimental observations), h is the groundwater level, and h_0 some equilibrium level of the groundwater table. The parameters a and b account for the effects of soil hydraulic properties (of the deeper layers) and regional flow on changes in the groundwater table.

The nodal flux at each time during the simulation is calculated using Equation 1.1. This flux depends upon the pressure head, which varies during the simulation. You are required to enter three values:

- o The parameters a and b (of Equation 1.1).
- o The reference position of the groundwater table, h_0 , which is like a datum value.

The calculated flux is assigned to the node at each time step. Applications of this BC are limited because it requires field data to evaluate the fitting parameters.

8. Seepage Face: this is a dynamic outflow (or drainage) BC that changes according to the flow conditions during the simulation. The user selects the potential seepage nodes through which water may seep out of the domain during the simulation. HYDRUS assumes a uniform pressure head equal to zero along the saturated (active) part of the seepage face through which water seeps out from the saturated part of the domain. Along the unsaturated segment of the seepage face, where water is still held up in the capillaries (under tension), the flux across the boundary is assumed to be zero. Since the length of the active seepage face is dynamic (i.e., it changes during the simulation), the active seepage nodes may decrease or increase during any time step. This forces the numerical solution to be iterative.

A seepage face occurs in many cases like:

- Water freely draining through a soil surface exposed to the outer atmosphere, like the open bottom boundary of a finite soil column, or to a tile drain.

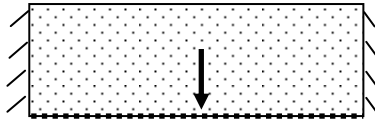


Figure 1.14: Seepage across the bottom of an open-ended finite soil

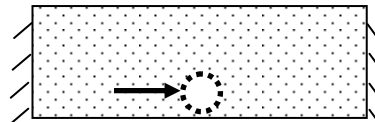


Figure 1.15: Seepage into a tile drain
Example: Direct\Furrow

- Water draining through a soil surface in contact with a filter material having a large pore structure like gravels or coarse sands as in ditch drains.

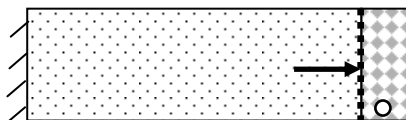


Figure 1.16: Seepage across the face of a ditch drain

- Seepage occurs at the downstream side of a dam where water seeps above the phreatic level.

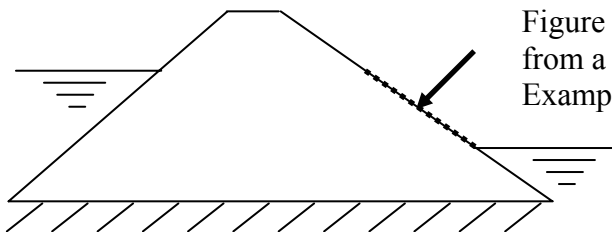


Figure 1.17: Seepage from a dam
Example: Direct1\Dike

- Water may seep through an unlined tunnel.

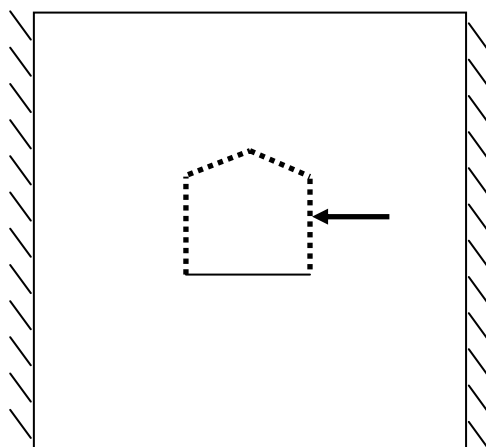
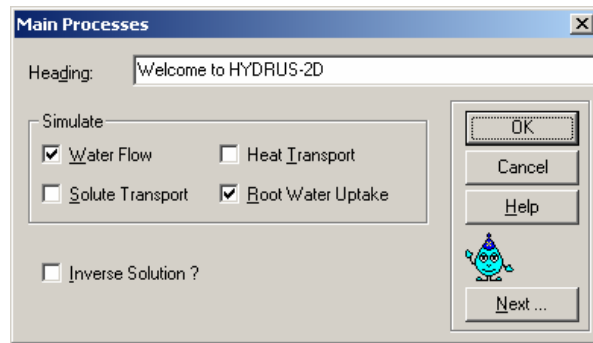


Figure 1.18: Seepage into an underground tunnel
Example: Direct1\Tunnel

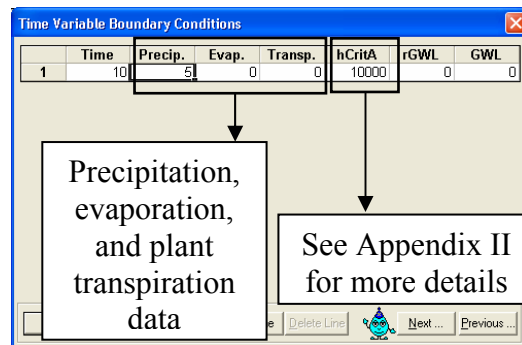
9. Atmospheric Boundary

Condition: this boundary condition lets us incorporate climatic conditions like rainfall (precipitation) and evaporation, or transpiration (root uptake) by plants. The latter is only activated if root water uptake is checked in ‘Main Processes’ as shown.



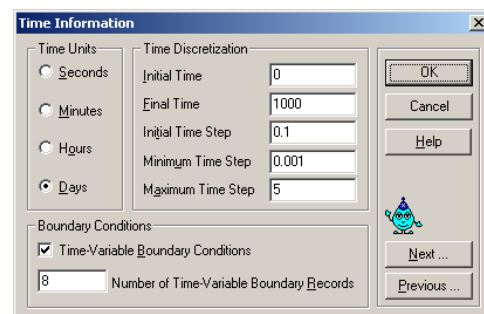
The window shown here will appear only if ‘Time-Variable BC’ is checked in the ‘Time Information’ window.

The data are entered as a time series. The number of records in the series depends on the variability of the data, not the length of the simulation. For example, you may have a 10-day simulation where the precipitation is recorded at an average rate of 5 cm/day throughout the 10-day period, in which case you will need only one record as shown here.



However, you may also have rainfall data obtained from a 0.5 mm tipping bucket for a 12-hour duration. The raw data are shown in columns 1 and 2 of Table 1.1. Precipitation in HYDRUS is entered as a rate (having the units chosen for the simulation, cm/day in this example). Using the tipping-bucket data, we must calculate a rate during each tipping period as follows (refer to Table 1.1):

- Column 3: calculate the duration between consecutive tips. Convert ‘min’ to ‘day’ in Column 4.
- Column 5: convert rainfall to cm.
- Column 6: calculate cumulative time (simulation time).
- Column 7: calculate rainfall rate, Column5/Column4
- Columns 6 and 7 are ready to be entered as ‘Time Variable Boundary Conditions’ under ‘Time’ and ‘Precip.’. In many cases (especially for very long simulations) the weather data are best entered from a spreadsheet. The easiest way to enter the data is as follows:
 - Enter the exact number of Time-Variable Boundary Records, 8 (Number of Time-Variable Boundary Records) in this case.



- Copy the 8 cells from Columns 6 and 7 (shaded cells in Table 1.1).
- Open ‘Variable Boundary Conditions’; the table should contain 8 rows. Select the first cell in Row 1 under column ‘Time’, and Press “Ctrl_v” on the keyboard; the data should appear as in the Window shown here.

	Time	Precip.	Evap.	Transp.	hCritA	rGWL	GWL
1	0.0042	24	0	0	10000	0	0
2	0.0083	12	0	0	10000	0	0
3	0.0167	6	0	0	10000	0	0
4	0.025	6	0	0	10000	0	0
5	0.0417	3	0	0	10000	0	0
6	0.1125	0.7059	0	0	10000	0	0
7	0.35	0.2105	0	0	10000	0	0
8	0.5	0.3333	0	0	10000	0	0

Table 1.1: Rainfall data

Time	Rainfall (mm)	Duration (min)	Duration (day)	Rainfall (cm)	Time (day) Cumulative	Rate (cm/day)
5:36:00						
5:42:00	1	6	0.00417	0.1	0.0042	24
5:48:00	0.5	6	0.00417	0.05	0.0083	12
6:00:00	0.5	12	0.00833	0.05	0.0167	6
6:12:00	0.5	12	0.00833	0.05	0.0250	6
6:36:00	0.5	24	0.01667	0.05	0.0417	3
8:18:00	0.5	102	0.07083	0.05	0.1125	0.7059
14:00:00	0.5	342	0.23750	0.05	0.3500	0.2105
17:36:00	0.5	216	0.15000	0.05	0.5000	0.3333

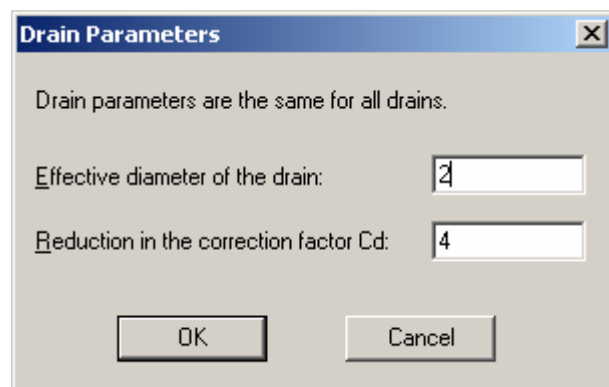
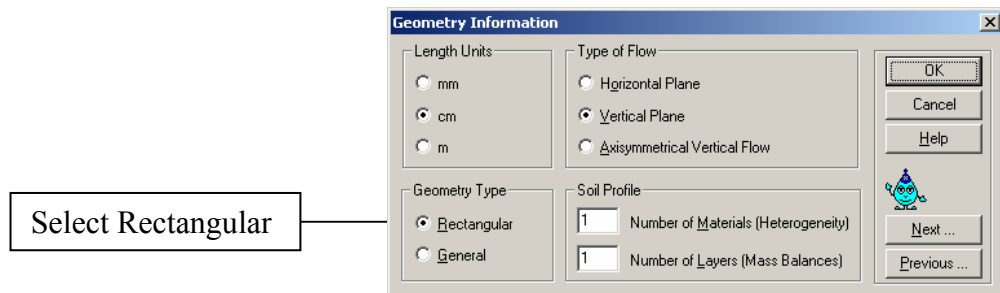
What if the specified flux is higher than what the soil can accommodate?

When an atmospheric flux boundary condition is used and the specified flux into the soil is higher than the infiltration capacity of the soil, HYDRUS switches the BC from a flux to a pressure head BC, with the pressure head then becoming zero along that boundary. The amount of water infiltrating the surface is subsequently calculated according to the hydraulic conductivity and pressure head gradient at the soil surface (using Darcy’s law), while excess water is instantly removed as surface runoff. Similarly, when the soil cannot supply the specified flux out of the soil, some generally low (negative) constant pressure head (h_{CritA}) is specified, and the flux out of the profile is calculated.

How many records can one enter using time-variable boundary conditions?

There is no limit on the number of time-variable BCs in the computational module. The interface, however, is limited to 15,000 data records. If you feel it is difficult to manually enter thousands of data or want to use more than 15,000 you can enter the data directly into the “atmosph.in” input file using a spreadsheet. Specify the total number under ‘MaxAI’, then enter as many columns as needed.

10. Drain Boundary Condition: this boundary condition is only available for use with 'Rectangular Geometry' (if you select 'General' in 'Geometry Information', the 'Drain' option in the 'Boundary Condition Editor' will be inactive).

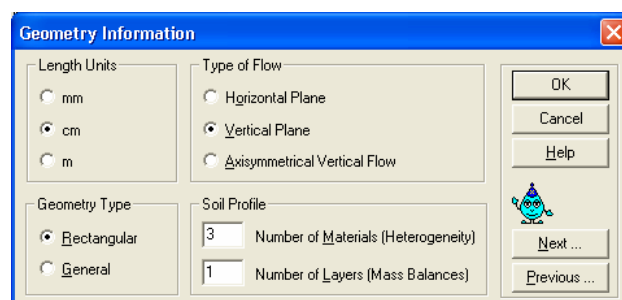


Boundary Conditions/Conditions/Drains/Insert 

When you are finished inserting drains, click the right mouse button, and enter parameters in this window. This option implements a drain by means of an equivalent nodal sink. Refer to example (Direct1/drainage) included in the HYDRUS package.

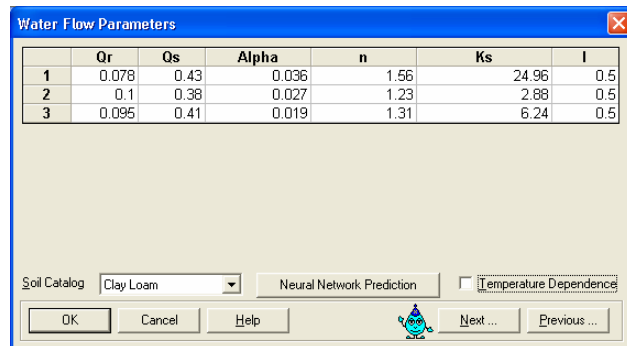
1.1.10.2 Material Distribution

HYDRUS defaults to a homogeneous domain having only one material (with one set of material properties). However, you can model heterogeneous domains by introducing more than one material. This is done as follows:



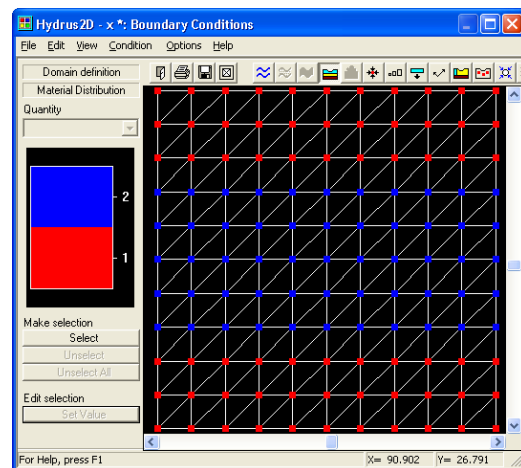
In “Geometry Information, Soil Profile”: enter the number of materials that best represents your field conditions (up to 100). We selected here 3 materials.

In “Water Flow Parameters”, you will see now a number of rows equal to the number of materials that you specified, 3 in this case. You should enter the water flow parameters for these soils.



	Qr	Qs	Alpha	n	Ks	l
1	0.078	0.43	0.036	1.56	24.96	0.5
2	0.1	0.38	0.027	1.23	2.88	0.5
3	0.095	0.41	0.019	1.31	6.24	0.5

In the “Boundary Condition Editor”, select “Condition/Material Distribution” or click the relevant button on the toolbar (🇺🇸). All of the domain defaults to Material 1. Select the nodes that you want to change. The “Edit Selection, Set Value” button on the sidebar will then be activated; click and enter the material number. Repeat for the third material. You will see the colour legend in the LHS of the window. Make sure to set the right number of materials before closing the window; otherwise a warning will be issued and you will not be allowed to leave the Boundary Conditions Editor until you do so.



For problems having a sloping soil profile (like Project “Mesh-2”), selecting the appropriate nodes with an orthogonal window is not easy. You may want to make the selection along the slope of the profile at any appropriate angle (parallel to the slope of the grid). In the “Boundary Condition Editor” select Condition/Material Distribution (🇺🇸).

In this example, you want to select all nodes along the upper boundary. You need to use the command “Edit/Select by Rhomboid”

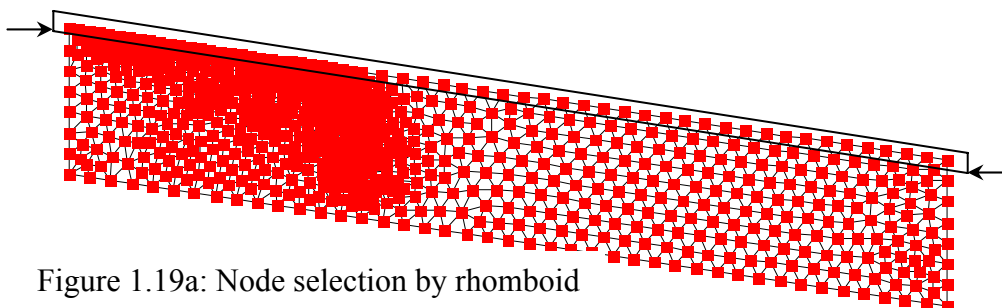


Figure 1.19a: Node selection by rhomboid

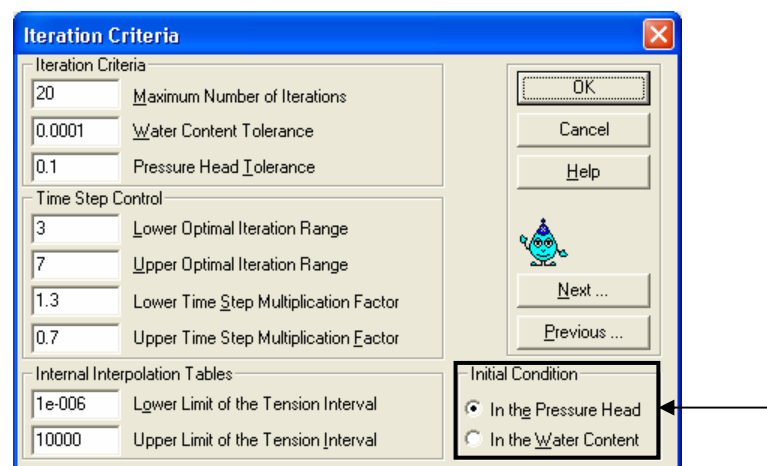
Click on the domain to define a line whose angle will set the orientation of the window as shown in the Figure 1.19a (the two points defined by the two arrows). Click and drag, and you will see that the selection window is no longer orthogonal.

1.1.10.3 Root Distribution

For all information related to root water uptake, refer to Section 3 and Appendix III.

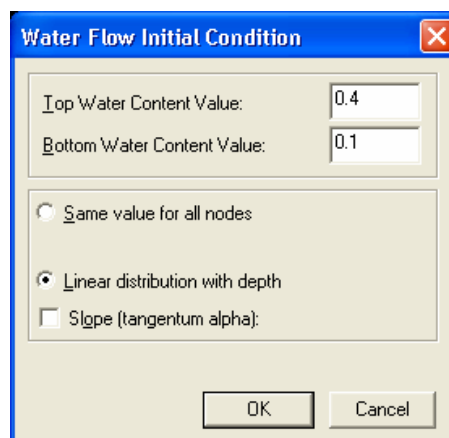
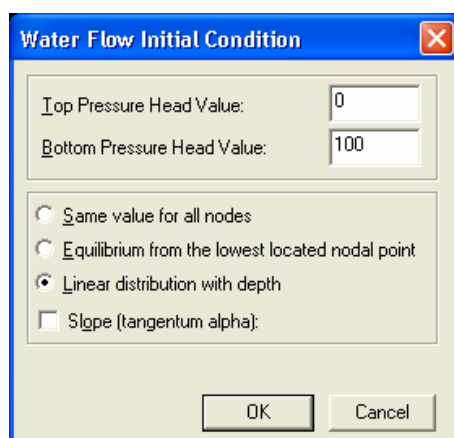
1.1.10.4 Initial Conditions

These refer to the initial soil conditions prior to the simulation. They could either be specified in terms of the water content or the pressure head, depending upon available data such as water contents from neutron probes/TDR probes, pressure heads from tensiometers (unsaturated zone), or pressure heads from pressure transducers (saturated zone). The values entered in the BC editor will be interpreted as water contents or pressure heads depending upon which box you selected under “Iteration Criteria, Initial Condition” as shown below.

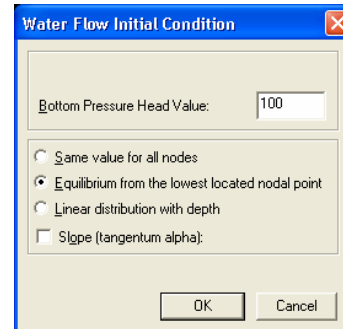
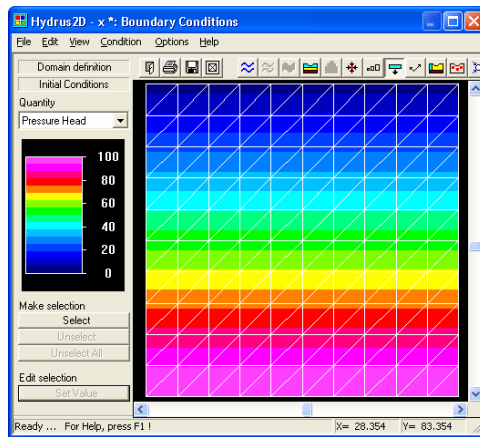


Select the relevant nodes. Either assign values to individual nodes or select the entire domain and choose one of the following four options:

1. Assign equal values to all nodes (for both water contents and pressure heads).
2. Linear distribution (for both water contents and pressure heads); specify the top and bottom values; HYDRUS will then linearly interpolate between them. This option is useful when imposing linearly variable gradients that are different from hydrostatic (not equal to 1; see Section 6.5).



- Assign hydrostatic equilibrium conditions (for pressure heads only); you have to specify the pressure head at the lowest point, and HYDRUS will calculate the rest. For example, if you have a 100-cm profile and you want the water table to be at the surface, enter 'Bottom Pressure Head Value'=100 (see below).



- When we model a hill slope and want to locate the water table at the surface of the slope, do the following (refer to Project 'Mesh-2'):

- Select 'Equilibrium From the Lowest Located Nodal Point'
- Check 'Slope (tangent alpha)'
- Enter the slope (tangent of angle); the slope in the figure below represents a negative tangent (the sign merely reflects the orientation of the slope).

The resulting head distribution is shown below.

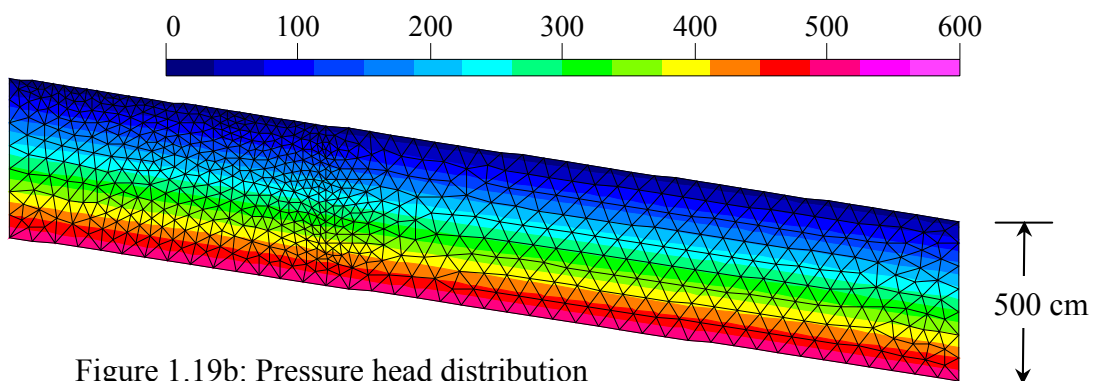
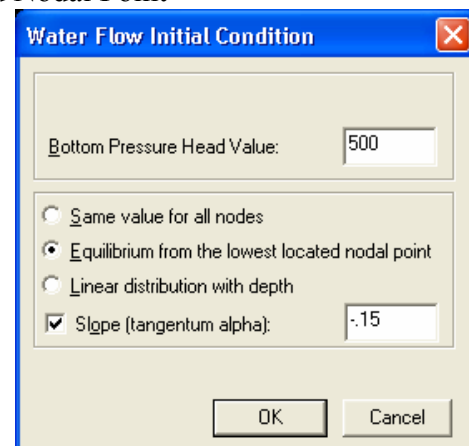


Figure 1.19b: Pressure head distribution

Another possible option is to import initial conditions (water contents or pressure heads) from a previous simulation, provided that problem had the same FE mesh. You may import the pressure heads for all nodes from any previous print time; HYDRUS then accesses the output ‘h.out’ file and prompts you to enter the time level. This option is useful in many cases, for example:

- Starting a simulation from a steady-state condition (will be demonstrated in this section).
- The boundary condition could change as in the case of a stream with a fluctuating water level (refer to Section 4.1 for an example).

Establishing a Steady-State Solution:

HYDRUS-2D can find the steady state flow profile using two approaches. First, one can specify initial and boundary conditions and run the program for a long time until pressure heads and/or water contents do not change anymore. Second, one can try to find the steady state solution in a single step by unselecting “Water Flow” in the “Main Processes” window. We will demonstrate this option by means of the following example.

Refer to example ‘Dike-ST’ on your CD (..\HYDRUS Manual Projects\Sect-1).

This example represents a dike with a hydrostatic constant pressure distribution along the LHS ($h=50$ m at toe), and a seepage face on the RHS (see Figure 1.20a). Since the initial condition shouldn’t matter that much, we have chosen equal pressure heads at all nodes, $h=0$. The problem is similar to ‘Direct1\Dike’ included in the HYDRUS examples.

Note that the steady-state solution is more suited for fully saturated conditions. It can also be found for unsaturated conditions when the soil parameters are not highly nonlinear and the initial conditions are close to the final steady state. In cases where the steady-state conditions could not be established, run a transient simulation for a long time until no changes are detected in the solution (can be done by monitoring observation nodes).

To obtain a steady-state solution, un-check all options in the ‘Main Processes’ window.

Click ‘[Calculation/Run HYDRUS](#)’ .

The solution should be quick; a message will say ‘Steady state was reached after – iterations’.

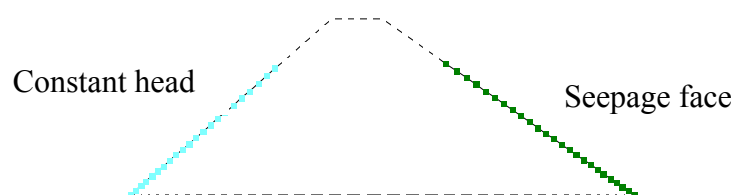


Figure 1.20a: Boundary conditions for Project dike

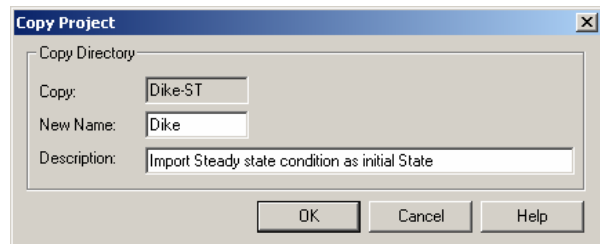
Import the Steady-State Solution

File/Project Manager

Select project 'Dike-ST'

Copy

Rename as in figure; in this way we obtain an identical copy of the project. We will import the steady-state solution as an initial state for the newly copied project 'Dike'.



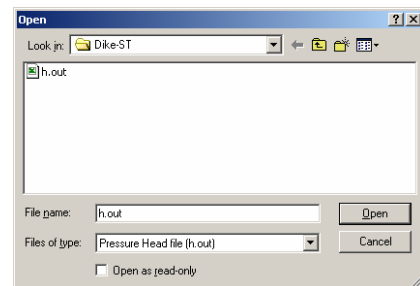
Boundary Condition Editor/Condition/Initial Condition/Import Pressure-Water Content

You will be prompted for a HYDRUS output file from a previous project

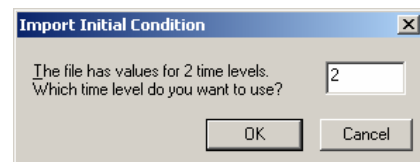
Browse and select the directory: 'Dike-ST'

Click on 'h.out' (since we are importing initial pressure heads)

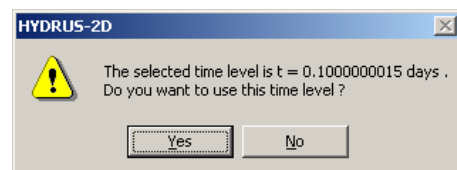
Open



This solution has only two time levels since it is a steady-state solution (initial and final steady state); print times hence are irrelevant. For other simulations (transient state) you have to select the time level you want to import as initial state (2 means the second print time). Click OK



Next you will be informed about the time at that print level. Select Yes.



The initial conditions for the model are shown in Figure 1.20b.

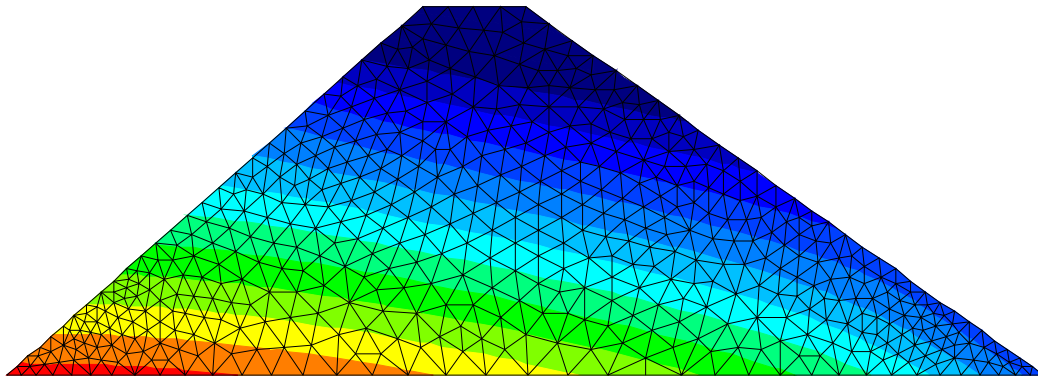
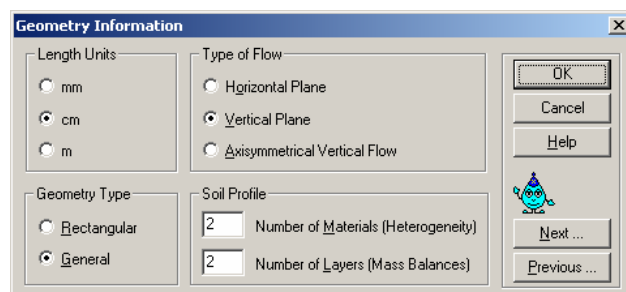


Figure 1.20b: Steady-state initial conditions

1.1.10.5 Sub-regions:

The domain may be subdivided into a number of sub-regions (maximum allowed is 10). Sub-regions do not affect the solution in any way but provide further mass balance information. The possible use of additional mass balance information is demonstrated in Section 2.2.3. For each sub-region, the amount of water in that sub-region and the sum of inflow/outflow is provided at each print time. This is how you specify sub-regions:

Geometry Information/Soil Profile/Number of Layers (Mass balances); enter the required number of sub-regions.



Select the area of each sub-region in a way similar to that used when selecting the 'Material Distribution'.


The user often wants the additional mass balance calculations when more than one material is present; HYDRUS can do this automatically. After finishing 'Material Distribution', simply click:

Options/Sub-region = Material Distribution. Manual adjustment is often needed since automatic assignment of subregions is not unambiguous; this is because materials are assigned to nodes and subregions to elements.

1.1.10.6 Scaling Factors

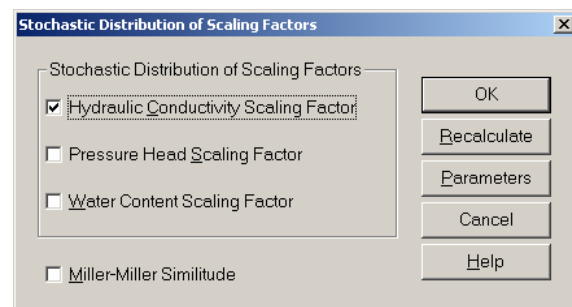
The concept of scaling factors is briefly described in Appendix IV. The following options are available in the “Boundary Conditions Editor”.

1. Explicitly assigning scaling factors to individual nodes

Boundary Conditions/Scaling Factors (or 

2. Automatic generation of scaling factors using stochastic distribution

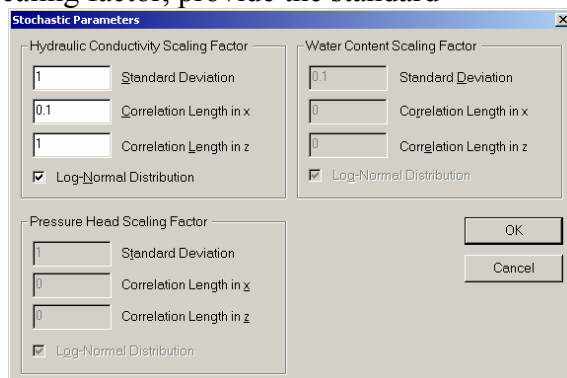
Options/Stochastic Distribution of Scaling Factors:



In addition to explicitly assigning scaling factor to grid nodes, there is an option of assigning stochastic (randomly distributed) scaling factors. This window appears when that option is selected. Select the parameters that will be scaled (check the relevant boxes).

The following window appears. For each scaling factor, provide the standard deviation, and the correlation lengths in the x and z directions. More details on the meaning of these parameters are found in Appendix IV.2.

When done, click OK. The previous window will re-appear; you can either click OK or view the results by clicking Re-calculate.



1.1.10.7 Observation Nodes

You may specify up to 10 nodes in the FE mesh to obtain a time (T-level) series of pressure heads, water contents, and solute concentrations.

Condition, Observation Nodes

Click Insert (sidebar buttons); cursor changes to an open square
Select the appropriate nodes.

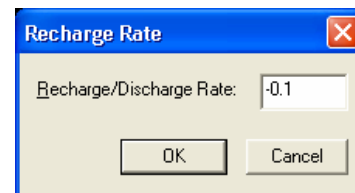
You can access the results from ‘Observation Points’ in ‘Post-Processing’.

1.1.10.8 Nodal Recharge

This option allows you to specify a sink/source node anywhere in the domain.

Condition, Nodal Recharge

Click Select (sidebar buttons); cursor changes to a hand; select the appropriate nodes.
Enter the rate as shown in the window here.



Note that a negative sign refers to a sink (water is being withdrawn from the domain). When a nodal recharge is specified to a set of external boundary nodes, HYDRUS considers those nodes to be constant flux nodes with the appropriate flux value. Refer to example 4.10 for a sample calculation. For 2-dimensional flow problems the units are L^2/T , and for 3-dimensional (axisymmetric) flow the units are L^3/T .

1.1.10.9 Local Anisotropy

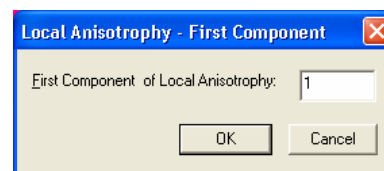
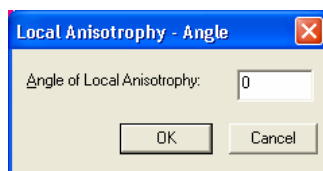
HYDRUS allows you to introduce anisotropy in the hydraulic conductivity as well as changes in the orientation of the local principal directions (in each element).

Condition, Local Anisotropy

Click Quantity (sidebar buttons); then sequentially select Angle, First Component (corresponds to x-axis), and Second Component (corresponds to z-axis).

Click Select (sidebar buttons); cursor changes to a hand; select the appropriate elements.

Click Set Value (sidebar buttons), then enter the appropriate values as shown below. The default values shown below refer to isotropic conditions.



Refer to example 4.8. For a theoretical background on ‘Implementation of hydraulic conductivity anisotropy’, refer to Section 5.3.12 (p. 55; HYDRUS Technical Manual).








1.2 Post-Processing

There are many ways of viewing and/or manipulating the simulation results. Post – processing is done in the RHS of the default window.

1.2.1 Graphical Display of Results

Open Project ‘Mbal-2’ (CD\HYDRUS Manual Projects\Sect-1); Double click on Graphical Display of Results (or .

You have the following options in the drop-down menu (see Figure 1.21):

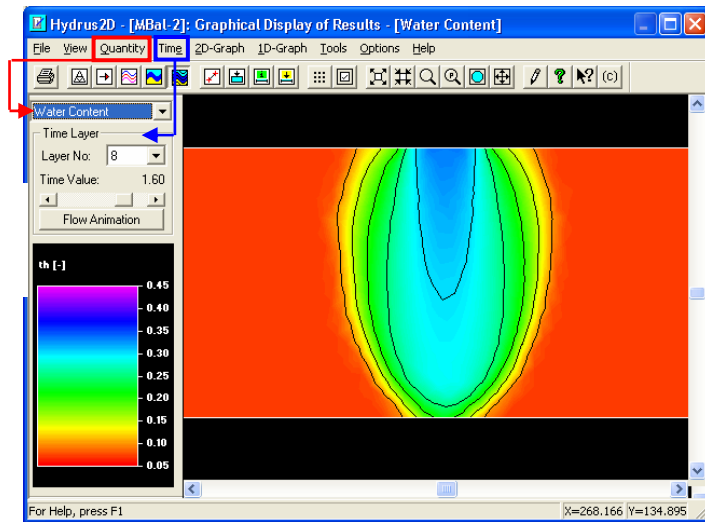
1. Quantity: Choose the variable you want to view; water content, pressure head, velocity, temperature, or concentration. You may choose them from the sidebar or the drop-down menu.
2. Time: Specify the print time you want to view. You may choose this option from the drop-down menu (Time/First (Next, Previous, Last) Time Level) or from the sidebar menu (scroll bar above Flow Animation button or from the drop-down list box – ‘Layer No:’).
3. Flow Animation: Having selected the variable to be displayed (e.g., head), click ‘Flow Animation’ to view an animated flow simulation.
4. 2D-Graph: specify the preferred visualization method, such as isolines () , spectrum () , or a combination of isolines and spectrum (.
5. 1D-Graph (inactive when velocity vectors are chosen in the 2D-Graph): You can get 1-dimensional plots along boundaries (, , ) , or along specified lines through the domain (.
6. Tools: You can search for a node, an element edge, or an element by entering its number.
7. Options: You may change display options such as isoline parameters, velocity vector parameters, and colour palettes.

All of these options are straightforward; we will demonstrate several of them here.

Screen Captures:

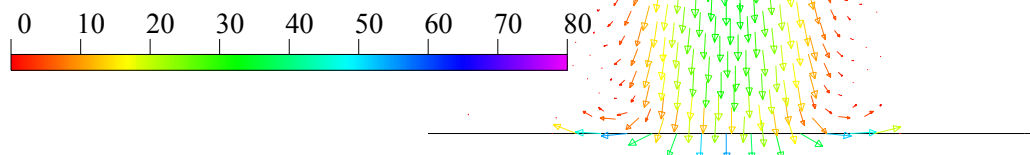
- You may capture the active window by pressing Alt-Shift-Print Screen, then pasting this window into another software like Word (Figure 1.21).
- You may capture the entire screen by pressing Shift-Print Screen
- Another option is to let HYDRUS do the screen capture for you. From 2-D Graphs, choose velocity vector. Zoom in on the area of interest in the middle. Go to: File/Copy to Clipboard, and choose view to obtain the velocity vector. Now you can paste the captured view into other softwares (e.g. MS Word, MS PowerPoint). Then repeat and choose spectrum. The results are shown in Figure 1.22.

Figure 21: Graphical Interphase



- Another option is to let HYDRUS do the screen capture for you. From 2-D Graphs, choose velocity vector. Zoom in on the area of interest in the middle. Go to: File/Copy to Clipboard, and choose view to obtain the velocity vector. Now you can paste captured view into other softwares (e.g. MS Word, MS PowerPoint). Then repeat and choose spectrum. The results are shown in Figure 1.22.

Figure 22: Velocity vectors for Project 'Mbal-2'



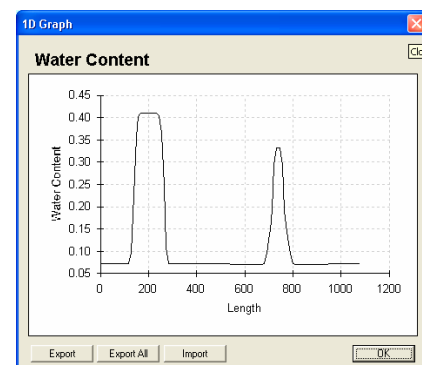
1-D Plots Along an External Boundary:

- You can obtain 1-D plots of any variable (water content, etc.) over the entire boundary. Try this:

Quantity/Water Content

1-D Graph/Boundary - Whole Curve (🖱️); the cursor turns into a hand. Click anywhere on the outer boundary of the domain.

You will obtain a plot as shown below. Note that 'Length' (x-axis) represents the perimeter of the external boundary of the domain.

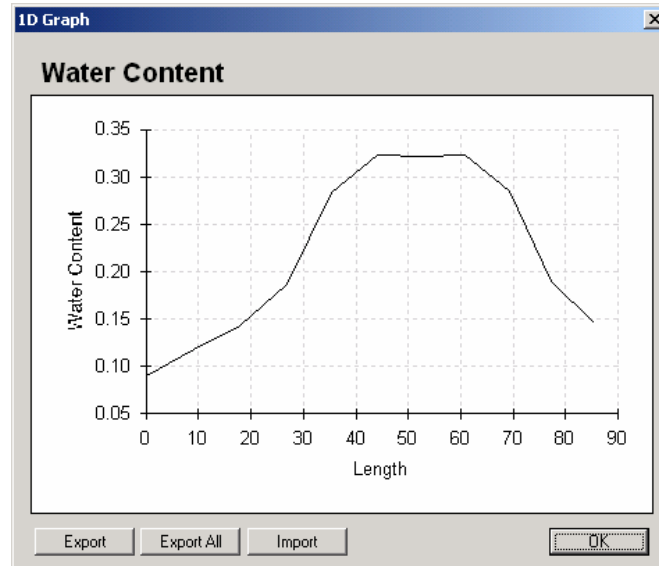



You can get 1-D plots of any variable (water content, etc.) over part of the boundary. Try this:

1-D Graph/Boundary Selection

Click and drag on the boundary part you want the plot for (this part will turn red), and click again to finish.

For the plot below, we dragged over the top boundary nodes (Y=140, X=150 to 230).

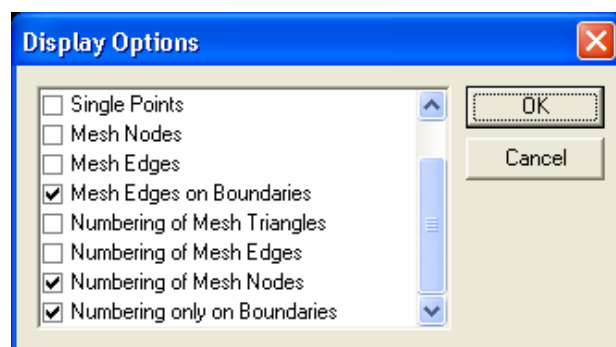


Alternatively, you can obtain the same information by selecting 'Boundary Indexes' . When using this option you need to know the node numbers that define the starting and ending points of the boundary section you want to plot. To find the node numbers, do the following:

Options/Display Options

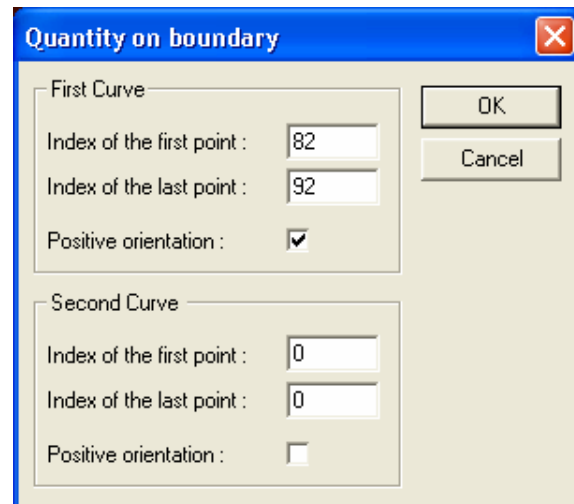
To get the node numbers only on the outer boundary, check the last two boxes as shown in the figure

You will see that the boundary section of interest is between nodes 82-92 (coordinates X=150-230).



1-D Graph/Boundary Indexes

Enter the values as shown in the figure; you will obtain the same 1-D plot for the defined portion of the boundary.

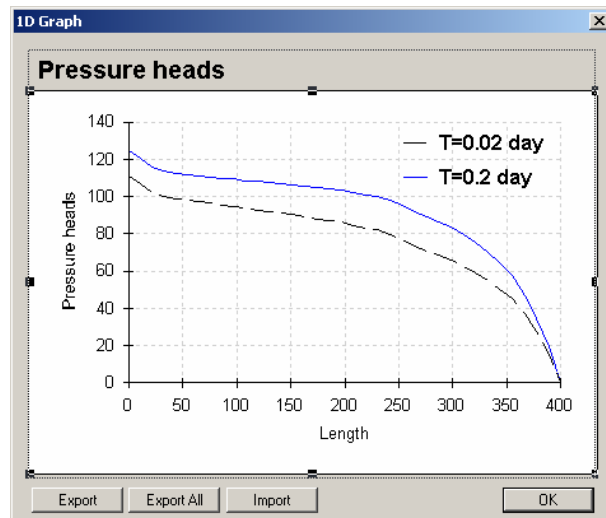


The dialog box 'Quantity on boundary' has two sections: 'First Curve' and 'Second Curve'.
 First Curve:
 Index of the first point: 82
 Index of the last point: 92
 Positive orientation:
 Second Curve:
 Index of the first point: 0
 Index of the last point: 0
 Positive orientation:

Cross-Sections along Outer Boundaries

We will demonstrate the use of cross-sections along the lower boundary of the domain for Project 'Mbal-3'. This option will allow you to view the water table profile at two different times during drainage.

1. Open Project (CD\HYDRUS Manual Projects\Sect-2\Mbal-3).
2. Double click on Graphical Display of Results
3. Quantity/Pressure Heads
4. Select Time Layer 10, or select Time Value = 0.15 day
5. 1_D Graph/Boundary Selection/Select the lower boundary of the domain
6. Export/give a file name/save (CD\HYDRUS Manual Projects\Sect-2\head_T_value_10)
7. Repeat step 5 for Time Value=0.2 day
8. Import/enter the file name you saved for time=0.02 day
9. Double click on a line and change its style to dashed.
10. To add legends
 Right-click on the plot
 Legend/Visible, OK
 Right-click on the plot
 Edit chart data
 Click on column heading (c1) and write T=0.02 day; repeat for column (c4) and write T=0.2 day



The resulting figure shows the water table profiles at two times during drainage.

Cross-Sections Through the Domain

It is possible to get a cross section through the domain at any angle.

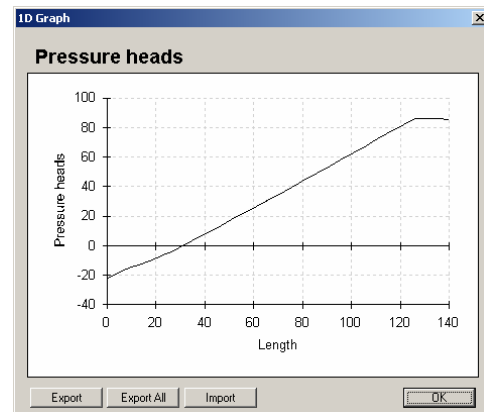
Here we demonstrate a vertical cross-section that shows the vertical pressure head profile.

The cross-section shown here is at $T=0.2$ day; $x=200$.

1-D Graph/Cross Section


Click a point on the upper boundary.

To ensure a perfect vertical section, use the keyboard, use the 'Page Down' key for coarse downward movement then use the arrow keys to move the cross hairs vertically to a point exactly on the lower boundary.

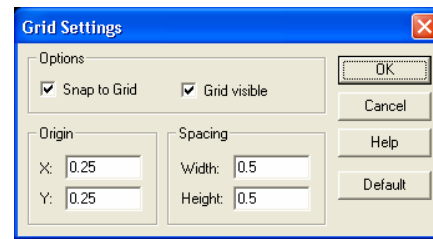


Alternatively, use the Snap to Grid option as described below.

If you are comparing cross-sections from consecutive runs, make sure that the cross-sections are identical, that is, taken at exactly the same position.

Under Options/Grid Settings , check both boxes "Snap to Grid" and "Grid Visible".

Change the origin of the grid, and choose suitable grid spacings (Width and Height) to ensure that the cross-section's starting and ending points coincide with a grid point so that the cross hairs will snap to it.



Velocity profiles can be generated also inside the domain, and then integrated externally to obtain flow rates across a cross-section inside the domain; refer to Section 4.5 for details.

Other Charting Options

By right clicking on the plot, you will obtain a long list of charting options. We will not discuss these in detail (some are actually disabled).

A few examples:

Edit Chart Data: get access to data values

Axis/Titles: changes titles, fonts, etc

Legend/Visible: adds a legend

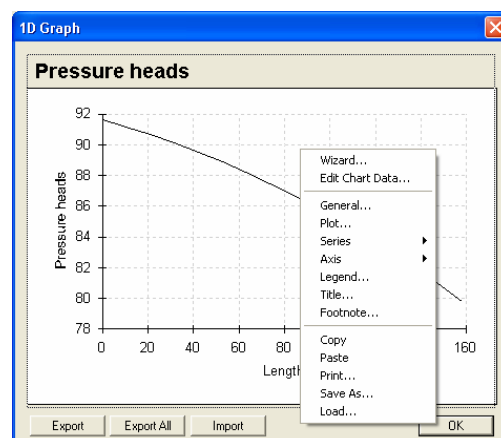
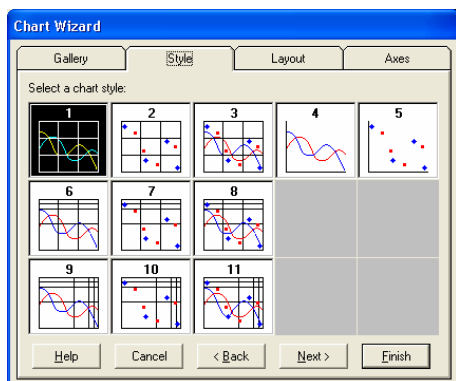
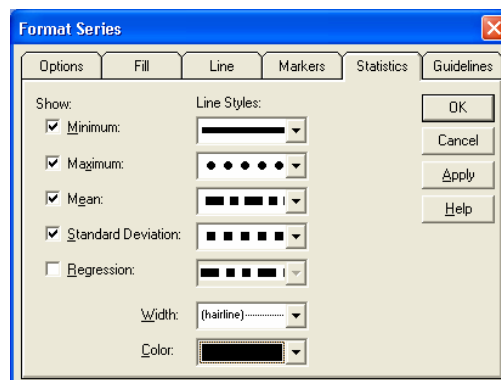


Chart Wizard gives these options:

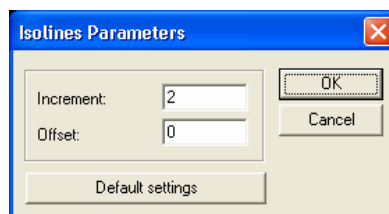


Format series allows you to obtain statistical parameters.

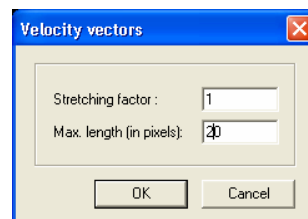


Other Features in Graphical Display of Results:

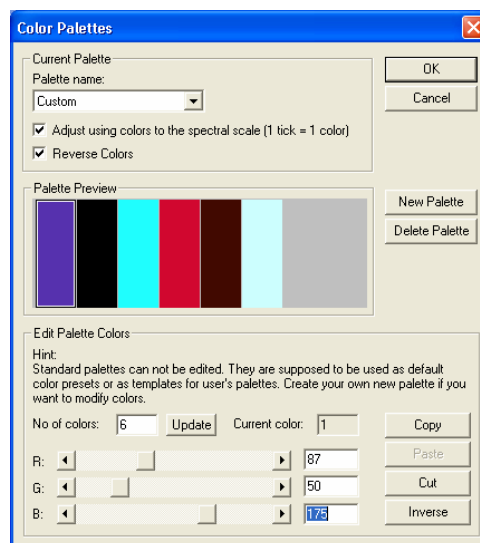
Options/Isoline Parameters: Allows you to change isoline settings, their density and starting point. A very small offset is sometimes useful to remove extremely small numerical oscillations from concentration contours.



Options/Velocity Vector Parameters: Allows you to change the size and scale of the velocity vectors.



Options/Colour Palette: Allows you to choose from many colours, or black-and-white palettes, or to custom-make your own. To make your own palette, select "Custom" from the Palette name. Enter a number in "No. of colors" (6 here). Click on each colour in "Palette Preview" then change the colour by scrolling on R, G, and B (red, green, and blue). Move to the next colour.



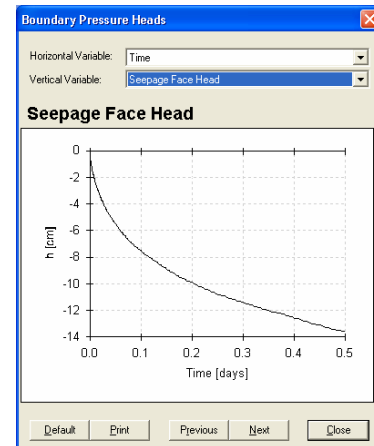
1.2.2 Pressure Heads

The data are found in the output file “h_mean.out”.

You get a time series of average pressure heads across boundaries.

Unit System:

Pressure heads are reported in length units.



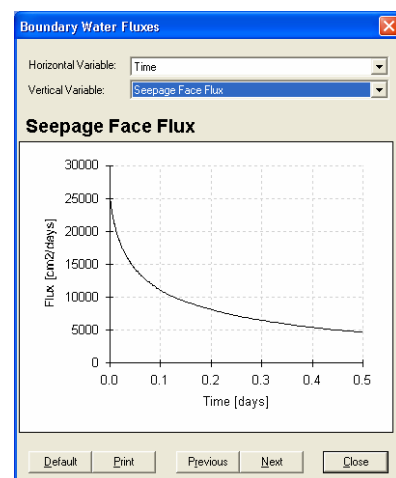
1.2.3 Water Boundary Fluxes

The data are found in the output file “v_mean.out”.

You get a time series of various boundary fluxes.

Unit System:

Atmospheric and root uptake (transpiration): L/T (length / time); the flux is reported per unit length of the boundary; like precipitation units (e.g., mm/day). It physically means: water volume/unit length of boundary/unit length in 3rd dimension/unit time ($\text{mm}^3/\text{mm}/\text{mm}/\text{day}=\text{mm}/\text{day}$). The 3rd dimension is perpendicular to the 2-D domain that we model.



Remaining (free drainage, seepage, etc.): V/T (area / time); the flux is reported over the entire boundary (volume/unit length in 3rd dimension/time). Note that V/T is the notation used in the output files. V has units of L^2 for two-dimensional problems and L^3 for axisymmetrical three-dimensional problems.

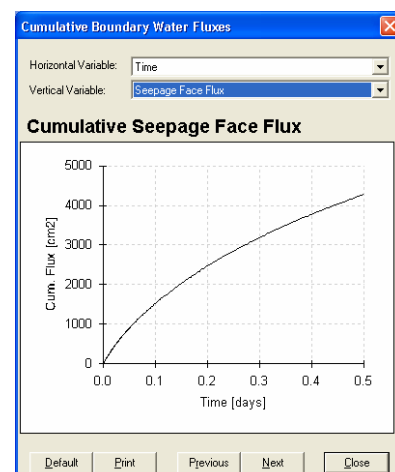
1.2.4 Cumulative Water Boundary Fluxes

The data are found in the output file “Cum_Q.out”.

Unit system:

Cumulative flow is reported in units of area (V) for 2D problems, which means volume/unit length in the 3rd dimension (e.g., cm^3/cm), and volume for axisymmetrical 3D problems.

Cumulative boundary flows are obtained by



integrating boundary fluxes (from the output file v_mean.out) to obtain the total volume of water (Q). Refer to Section 2.2.2 for example calculation.

For atmospheric and root uptake boundary: we integrate (sum up) fluxes over the length of the boundary and over time to obtain the cumulative flow.

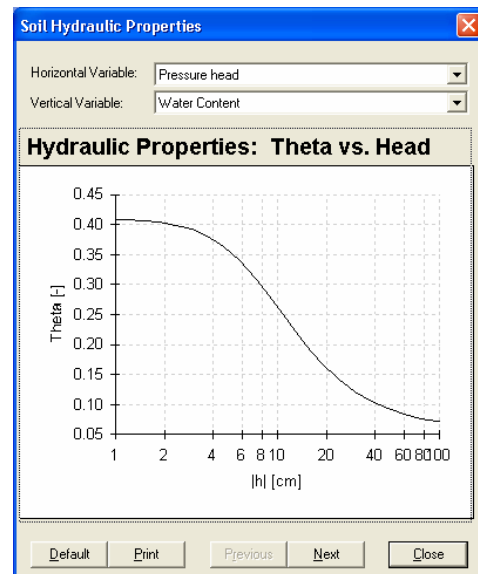
For other boundaries: we integrate fluxes over time to obtain cumulative flow.

1.2.5 Soil Hydraulic Properties

The data are found in the output file “check.out”

You can view the input soil hydraulic properties. For example, select pressure head for the x-axis and water content for the y-axis to obtain a plot of the water retention curve.

The retention curve is best plotted as water content versus pressure head on a logarithmic scale. Follow these steps:



After selecting the axes variables:

Right click on the chart

X-axis

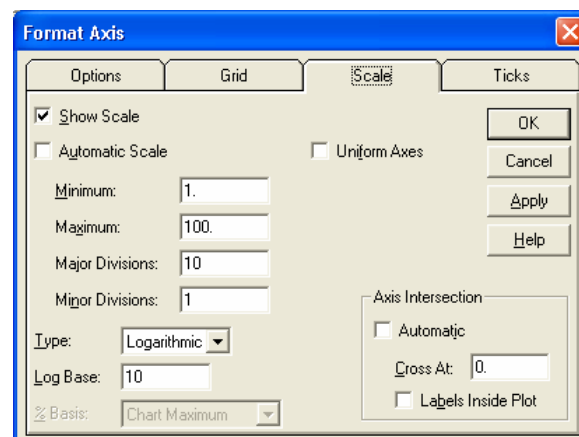
Value x-axis

Select

Change the values in “Format Axis” as shown.

Uncheck “Automatic scale” and enter shown values

Type: select “Logarithmic”

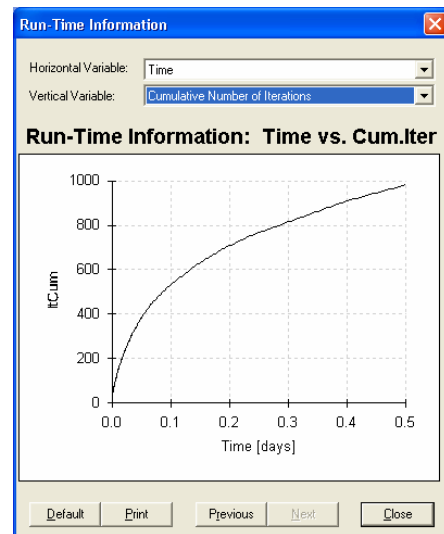


Alternatively, a log scale for pressure heads and conductivities is offered from the list boxes “Horizontal Variable” and “Vertical Variable”, respectively.

1.2.6 Run-Time Information

The data are found in the output file “Run_Inf.out”

Provides valuable run time information such as number of iterations at any time step. This feature allows you to check whether or not the maximum number of iterations has been exceeded at any time or not.



1.2.7 Mass Balance Information

Provides mass balance information for the entire domain and for individual sub-regions at print times.

Area: the area (2D) or volume (3D) of the domain (constant for all print times)

Volume: volume of soil water

InFlow: the sum of fluxes going into and leaving the domain (or sub-region).

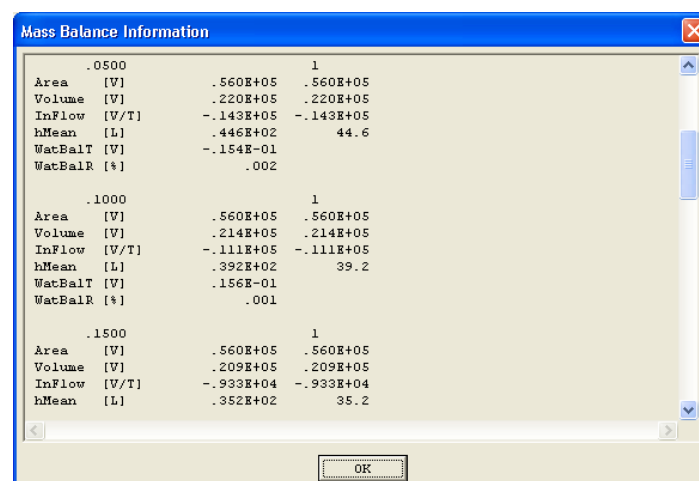
hMean: the average pressure head.

WatBalT (V): the total water balance error in the water volume of the domain.

WatBalR (%): the water balance error as a percentage.

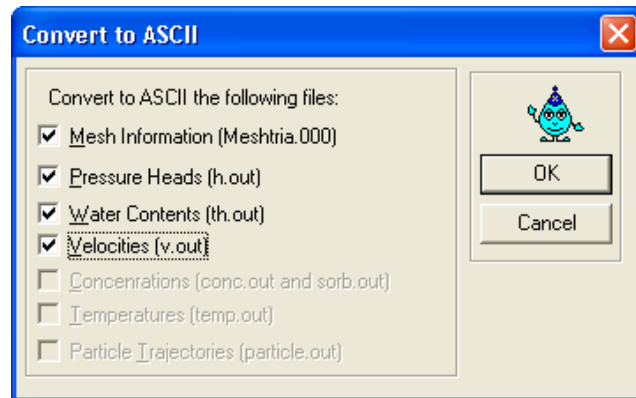
As an example, the screen capture below is for a drainage problem.

- Volume is decreasing with time because water is draining from the soil.
- Inflow is negative, indicating a net flux out of domain.
- Inflow decreases with time because drainage is faster at the beginning and then slows down.



1.2.8 Convert Binary Output to ASCII

During calculations HYDRUS generates the binary files 'h.out', 'th.out', 'v.out', 'concx.out', 'sorbx.out', and 'temp.out' that contain nodal values of pressure heads, water contents, velocities, solution and sorbed concentrations, and temperatures at selected print times. These binary files can be converted into ASCII files 'h.txt', 'th.txt', 'v.txt', 'concx.txt', 'sorbx.txt', and 'temp.txt' using the dialog window shown here.



The following information can be obtained in text format when needed. Check the appropriate boxes in the window; text files will be generated and saved in the Project directory. The files are:

1. TH.out -> Th.txt: water content
2. V.out -> v.txt: Velocities (in 'Print Information'; Print Fluxes box should be checked)
3. H.out -> h.out: Pressure heads

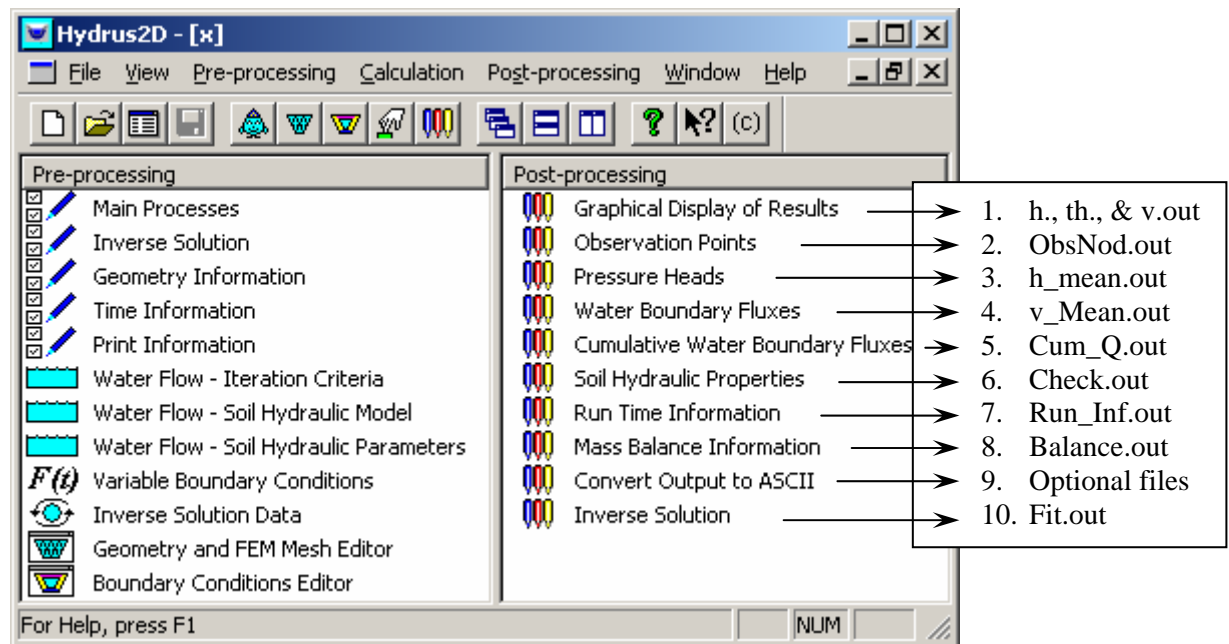
These three files provide nodal information at each print time. The data are reported in ten columns. That is, the first row represents data for nodes 1-10 and so forth. Output file 'V' has two data blocks at each print-time; one for the x-component and one for the z-component of the velocity. This information is used for animation, and for producing 1-D Graphs (along outer boundaries or internal cross-sections).

4. MeshTria.000 -> MeshTria.txt: mesh information

This file provides detailed mesh information by means of three data blocks; for Project 'Grid-1' Section 1.1.9.2; Example 1 (p. 1.18):

- First block: 1106 rows for nodes
- Second block: 3192 rows for edges
- Third block: 2086 rows for triangles

2. HYDRUS OUTPUT FILES



2.1 Output Files

The graphical display that appears in the post-processing section uses information from the output files in the manner shown above. These files are found in the local directory of the project. The units associated with the various variables are discussed in detail in Section 1.2.4. Note that 'P-Level' refers to print time-level, 'T-Level' to run-time, and 'A-Level' to a variable boundary condition time; refer to HYDRUS Technical Manual (p.161) for more details.

2.1.1 Boundary.out (data at print time; P-level information)

i	n	x	z	Code	Q**	V**	h	th	Temp	Conc
					[V/T]	[L/T]	[L]	[-]	[C]	[M/L ³]
Numbering of Boundary nodes	Number of node in FE mesh	Coordinates		As in Table 2.1	Nodal flow	Velocity (flux)	Head	Water content	Temperature	Concentration
Number of rows equal to number of outer boundary points (except for no-flux nodes)										
↓										

*The number of data blocks in each file is equal to the number of print times.

**You can access individual nodal fluxes and flow rates across an outer boundary at print times in this output file. The velocities reported in this file are more accurate than those used in the Graphical Module (obtained in ASCII form; see Section 2.1.9); they are the ones used for the mass balance calculations.

This file is stored in the project folder and is not displayed/used by the interface. For more details on the coding of boundary conditions (Table 2.1), refer to the HYDRUS Technical Manual (p.77).

Table 2.1: Boundary codes (column 5; 'boundary.out' output file)

1	Constant head	4	Atmospheric (capacity exceeded *)
-1	Constant Flux	-4	Atmospheric (capacity not exceeded)
-2	Potential Seepage (unsaturated)	-5	Drain (unsaturated)
2	Actual Seepage (saturated)	5	Drain (saturated)
3	Variable head	6	Free or Deep Drainage
-3	Variable Flux		

* Either the evaporation capacity exceeded (when head < hCritA), or the infiltration capacity exceeded (when head > hCritS).

2.1.2 ObsNod.out (data at run time; T-level information)

The output below shows that we have two observation points, and hence also two data blocks, for nodes 431 and 601, which refer to the numbers of observation nodes in the FE mesh. The information provided is pressure head (hNew), water content (theta), and temperature (Temp). This file exists only when you include observation nodes. Additional information on concentrations is displayed if solute transport is considered.

	Node(431)			Node(601)		
time	hNew	theta	Temp	hNew	theta	Temp
0	head	Water content	Temperature	-1	0.4293	20

2.1.3 h_mean.out (data at run time; T-level information)

This file contains average pressure heads along various boundaries, such as atmospheric boundary, root zone, variable (flux or head) boundaries, constant (flux or head) boundaries, seepage face, and drainage boundaries.

Time	HAtm	hRoot	hKode3	hKode1	hSeep	hKode5	hKode6
[T]	[L]	[L]	[L]	[L]	[L]	[L]	[L]
	Atmospheric	Root	Variable	Constant	Seepage	Drain	Free drainage

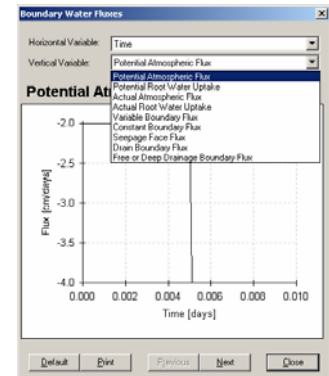
2.1.4 v_Mean.out (data at run time; T-level information)

This file contains average flux/flow rates across various boundaries, such as the potential atmospheric fluxes, potential root uptake fluxes, actual atmospheric fluxes, actual root uptake fluxes, variable (flux or head) boundary flow rates, constant (flux or head) boundary flow rates, seepage boundary flow rates, and drainage boundaries flow rates.

Note that the atmospheric and root data are reported as fluxes having units of length/time, whereas the other fluxes are flow rates reported as area/time (area represents volume per unit length in the perpendicular direction) for 2D problems. When calculating cumulative root and evaporation flow rates, the fluxes (in the 1st four columns) are integrated over the width of the atmospheric boundary condition.

Time	rAtm	rRoot	vAtm	vRoot	vKode3	vKode1	vSeep	vDrain	vBottom...
[T]	[L/T]	[L/T]	[L/T]	[L/T]	[V/T]	[V/T]	[V/T]	[V/T]	[V/T]
← Atmospheric boundaries →									
	Potential Atmospheric Rain+Evap	Potential Root Flux	Actual Atm. Flux	Actual Root Flux	Variable Boundary Flow rate	Constant Boundary Flow rate (constant head/flux & nodal recharge)	Seepage Flow rate	Drain Flow rate	Lower Boundary (free/deep drainage) Flow rate

Double click on “Boundary Water Fluxes/Vertical Variable”; the list that appears (shown here) corresponds to the columns in the “v_Mean.out” file (given above).



2.1.5 Cum_Q.out (data at run time; T-level information)

This file provides cumulative fluxes across various boundaries, that is, flow rates integrated over the simulation time. All values have area units (V; volume/unit length; e.g. cm³/cm=cm²) for 2D problems and volume units (cm³) for axisymmetric 3D problems.

Time	CumQAP	CumQRP	CumQA	CumQR	CumQ3	CumQ1	CumQS	CumQ5	CumQ6
[T]	[V]	[V]	[V]	[V]	[V]	[V]	[V]	[V]	[V]
	Potential Evaporation Flux	Potential Root Uptake Flux	Actual Evaporation Flux	Actual Root Uptake Flux	Cumulative Variable Boundary Flux	Cumulative Constant Boundary Flux	Cumulative Seepage Flux	Cumulative Drain Flow Flux	Cumulative Free/Deep Drainage Flux

2.1.6 Check.out

This file contains the data in the look-up table used by HYDRUS to interpolate soil hydraulic properties during the simulation, in addition to some of the input parameters contained in the input files. The information in this file may be used to plot water retention and hydraulic conductivity curves of the soil. The range of values for h, K, etc. depends on the limits entered in “Internal Interpolation Tables”, Section 1.1.5.

Mat	Qr	Qs	Alfa	n	Ks	l	
1	0.078	0.43	3.60E-02	1.56E+00	2.50E+01	0.5	
Table	of	Hydraulic	Properties	which	are	interpolated	
theta	h	Log (h)	C	K	Log(K)	l	

2.1.7 Run_Inf.out (data at run time; T-level information)

This file provides run-time information, which includes time level, time, time step, number of iterations, and cumulative number of time steps at every time step. The first column is simply a counter for time steps, always an integer whose final value depends on the minimum time step, the maximum time step, and how quickly or slowly the solution converges. Refer to the Chart 1.1 (p 1.9) on how the time stepping scheme proceeds. Additional information on the maximum nodal Peclet and Courant numbers is displayed if solute transport is considered.

TLevel	Time	dt	Iter	ItCum
Time-level 'current time step number'	Time at current time-level	Time step	Number of iterations	Cumulative number of iterations
1	1.00E-03	1.00E-03	2	2
2	2.30E-03	1.30E-03	2	4
3	3.99E-03	1.69E-03	2	6

2.1.8 Balance.out (data at print time; P-level information)

This file provides mass balance information at the print times. The sample file below is for a simulation with two print times (at 0.5 and 1 day). Initial conditions are always reported at the beginning of the file (Time=0).

The print time is printed at the beginning of each data set. The set comprises: Area, Volume, InFlow, hMean, and water balance errors for the whole domain and each subregion. Explanations of the terms are given below in the column 'Explanation'.

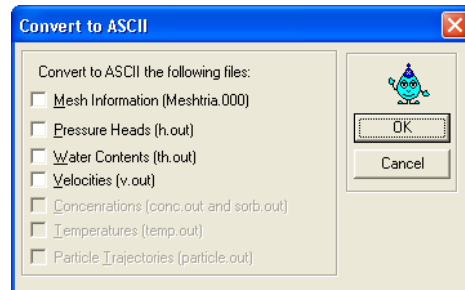
Time	[T]	Total	Sub-region	number	Explanation
					For total grid and each sub-region
0			1	2	
Area	[V]	1.92E-04	9.82E-05	9.38E-05	Area
Volume	[V]	8.24E-05	4.21E-05	4.03E-05	Water volume stored
InFlow	[V/T]	0.00E+00	0.00E+00	0.00E+00	Σ In/out flow (out is +)
hMean	[L]	-1.00E+00	-1	-1	Mean head
0.5			1	2	
Area	[V]	1.92E-04	9.82E-05	9.38E-05	
Volume	[V]	8.24E-05	4.21E-05	4.03E-05	
InFlow	[V/T]	0.00E+00	0.00E+00	0.00E+00	
hMean	[L]	-1.20E+00	-1.2	-1.2	
WatBalT	[V]	8.45E-07			Absolute error in water mass balance of entire domain
WatBalR	[%]	.025			Relative error in water mass balance of entire domain
1.0			1	2	
Area	[V]	1.92E-04	9.82E-05	9.38E-05	
Volume	[V]	8.24E-05	4.21E-05	4.03E-05	
InFlow	[V/T]	0.00E+00	0.00E+00	0.00E+00	
hMean	[L]	-1.20E+00	-1.2	-1.2	
WatBalT	[V]	1.64E-06			
WatBalR	[%]	.035			

2.1.9 Optional ASCII files (data at run time; P-level information; for files 'h.out', 'th.out', and 'v.out')

During the calculations HYDRUS generates the binary files 'h.out', 'th.out', 'v.out', 'concx.out', 'sorbx.out', and 'temp.out' that contain nodal values of pressure heads, water contents, velocities, solution and sorbed concentrations, and temperatures at selected print times. These binary files can be converted into the ASCII files 'h.txt', 'th.Txt', 'v. txt', 'concx.Txt', 'sorbx.Txt', and 'temp.txt' using the following dialog window. These files are not available by default. Once this option is triggered, this window is displayed.

The options are:

- Mesh Information: details finite element mesh information (file name; MeshTria.txt)
- Pressure head: nodal pressure heads at the print times (file name; H.txt).
- Water content: nodal water contents at the print times (file name; TH.txt).
- Velocities: x- and z-components of the nodal velocities at the print times (file name; V.txt).



More details can be found in Section 1.2.8 (P 1.53).

2.1.10 Fit.out

This file relates to the inverse solution. The file lists the input soil hydraulic parameters, the optimisation results at each iteration, the final optimized (estimated) parameters as obtained with a non-linear least squares analysis (final results are numbers shown in **Bold**), input inverse data (i.e., observed data that HYDRUS is calibrating against; found under column 'Obs'), and the simulated data (found under column 'Fitted'). The meaning of 'Type' and 'position' is explained in the HYDRUS on-line Help.

Non-linear	least-squares	analysis:	final	results			
	95% Confidence	limits					
Variable	Value	S.E.Coeff.	Lower	Upper			
ALPHA	1.56E-02	0.00082	0.01373	0.01738			
N	2.19E+00	0.10786	1.94905	2.42969			
CONDS	1.27E-02	0.00234	0.00748	0.01788			
RSQUARE	for	regression	of	predicted	vs	observed	=
Contributions	to	the	final	objective	function		
Measurement	Set	2:00	1.66E-01				
&	Fitted	QUANTITY					
Resi-							
No	Time	Obs	Fitted	dual	Type	Position	
1	33	0.113	0.112	0.001	2	1	
2	66	0.121	0.112	0.008	2	1	
3	100	0.142	0.126	0.016	2	1	

Observed data (experimental data; what you entered)

Fitted data (what HYDRUS simulated using the final optimized parameters)

Correlation matrix

	ALPHA	N	CONDS
ALPHA	1		
N	-0.2156	1	
CONDS	0.9887	-0.3591	1

The correlation matrix (shown above) indicates whether the parameters are correlated or not, with 1 indicating perfect positive correlation and –1 indicating perfect negative correlation.

S.E. Coeff. refers to the standard error.

RSQUARE refers to the goodness of fit.

Upper and lower limits refer to the sensitivity of the model to the parameter. Refer to Appendix V.2 for more details on statistical issues in inverse modelling.

2.1.11 A_Level.out (data at variable boundary time)

This file contains data at the variable boundary time levels, that is, the number of times (rows in this file) is equal to the time-variable boundary records entered in Window ‘Time Information/Boundary Conditions’. This file will not be created if you do not have time-variable boundary conditions (infiltration, evaporation, transpiration, variable flux, or variable ground water table).

All	cumulative	fluxes	(CumQ)	are	positive	out	of	the	region
Time	CumQAP	CumQRP	CumQA	CumQR	CumQ3	hAtm	hRoot	hKode3	A-level
[T]	[V]	[V]	[V]	[V]	[V]	[L]	[L]	[L]	
Time	Potential cumulative flow across atmospheric boundary	Cumulative potential root uptake (transpiration)	Actual cumulative flow across atmospheric boundary	Cumulative Actual root uptake	Cumulative flow across variable boundary	Average head at atmospheric boundary	Average head in root zone	Average head variable boundary	Number of ‘A’ Level

2.2 Examples Demonstrating the Use of Output Files

2.2.1 Project ‘WT-1’ (CD\HYDRUS Manual Projects\Sect-2): This example demonstrates a case where we must have access to the HYDRUS output files. This problem considers a fully saturated soil profile (i.e., water table at the soil surface). We simulate the decline of the water table as a result of evaporation (see Figure 2.1), and want to plot the actual surface flux ratio (actual over potential flux) versus depth to the water table. For this purpose we need to place an observation node at the base of the soil profile, and then plot the flux (from ‘v_Mean.out’ file) versus the pressure head (from the ‘ObsNod.out’ file).

NOTE: The initial water table is at the soil surface; however, experience has shown that setting the initial head at the surface to 0 is often problematic for getting convergence in the numerical solution (the entire domain is then saturated). The problem is avoided by setting the initial water table depth at 1 cm.

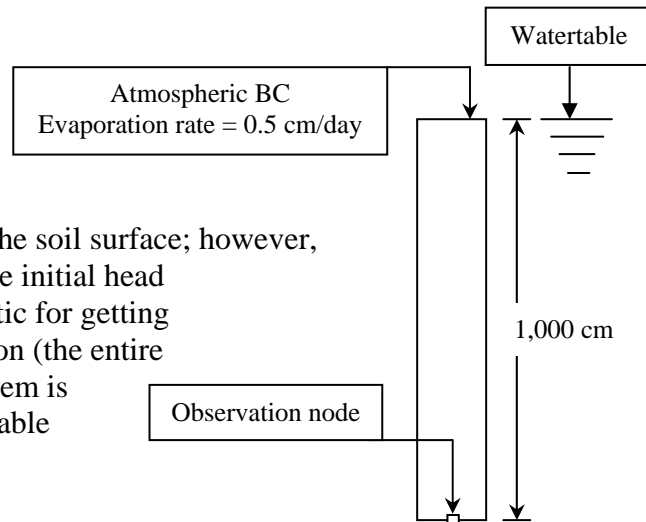


Figure 2.1: Domain and BC
Project WT-1

Open file **Flux Ratio-1.xls** on the CD (CD\HYDRUS Manual Projects\Sect-2).

The relevant output files are imported into two worksheets, 'v_Mean' and 'ObsNod'.

In order to obtain the flux ratio we normalise the flux with respect to the potential evaporation rate, that is, we divide all fluxes by 0.5 cm/d. The depth to the water table is calculated by subtracting the head from the initial head (remember 999 rather than 1000).

Since the data in both files are at the same run time levels, we can plot flux ratios calculated from the v_Mean.out file versus depth to water table calculated from 'ObsNod.out' file. The result is demonstrated in Figure 2.2, which shows the critical water table depth below which the soil cannot sustain the flux at its potential rate.

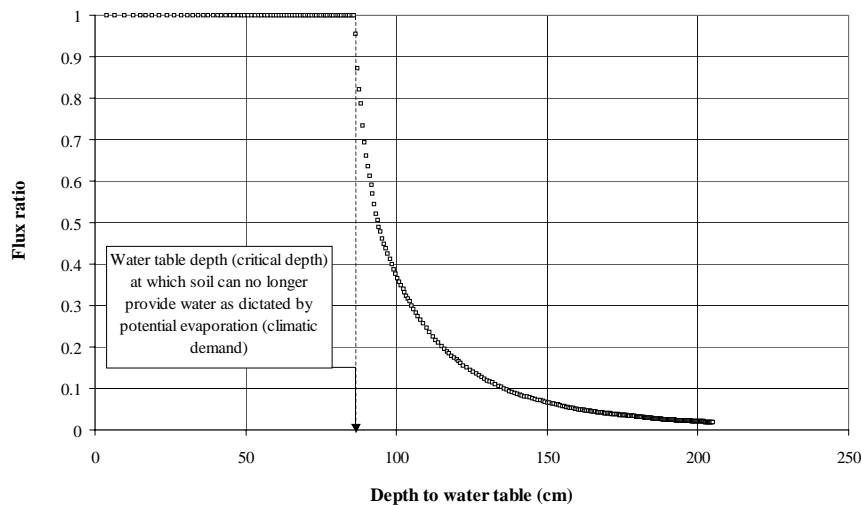


Figure 2.2: Change of flux ratio with water table depth

2.2.2 Project “Mbal-2” (CD\HYDRUS Manual Projects\Sect-2): the domain of the problem is shown in Figure 2.3. Part of the upper boundary is an atmospheric BC and the lower boundary is a seepage face; rainfall is simulated at two different rates during the 2-day simulation.

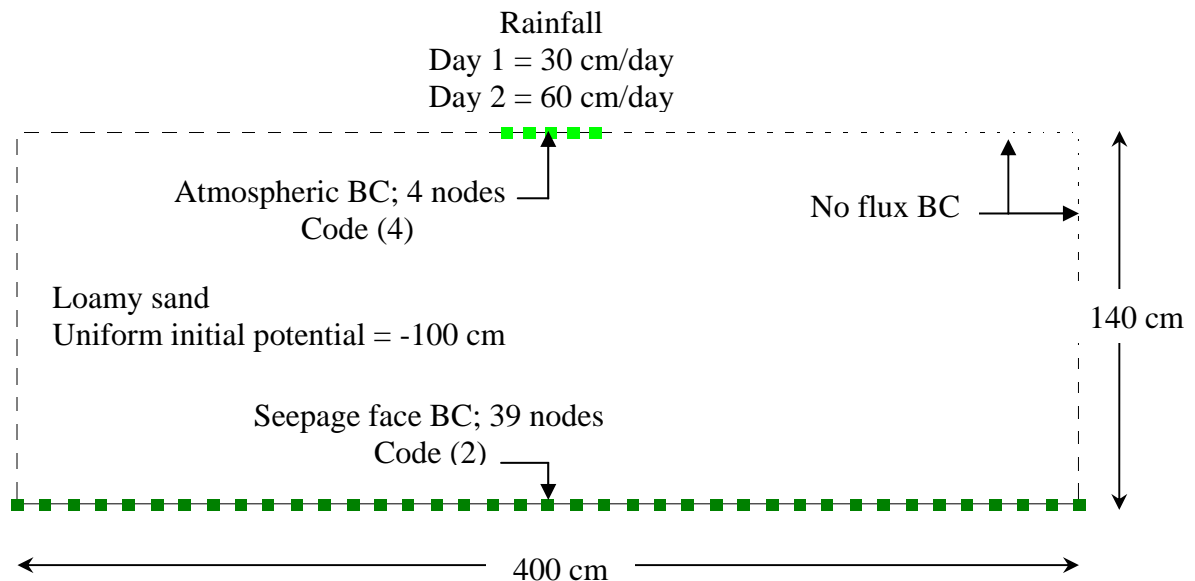


Figure 2.3: Domain and BCs for Project Mbal-2

We provide details by means of the following files:

1. Boundary.out
2. v_Mean.out
3. Cum_Q.out
4. A-Level
5. Balance.out

Those files contain the most valuable flux and mass balance data. We will demonstrate how the data in those files are interrelated.

Open file **Mbal-2.xls** on the CD (CD\HYDRUS Manual Projects\Sect-2).

1. **Boundary.out** data at print time level (in Worksheet: Boundary):

Column 1: $i = 1, 2, \dots, 44$ represents the boundary nodes (other than no-flux); in this problem the sum of seepage and atmospheric boundary nodes is 44 (39+5).

Column 2: n represents the actual corresponding node number in the FE mesh.

Columns 3, 4: x and z represent the coordinates of the nodes. For example, if you want to know the width of the atmospheric boundary condition, do the following:

$$x_{(I=44)} - x_{(I=40)} = 217.8 - 184.1 = \mathbf{33.7 \text{ cm}}$$

Column 5: code refers to the boundary condition code (as listed in Table 2.1). Note that all seepage face nodes are coded (-2) up to Time=1.6 day, which means that they are considered to be a potential seepage face (i.e., there is no actual seepage face yet). At Time=1.8 days, nodes (I=19-21) are coded (2), which means they are now actual seepage face nodes. At T=2 days, the seepage face extends to 5 nodes (I=18-22).

Column 7: v refers to the nodal flux (cm/day). For atmospheric BCs, v is simply the rate of rainfall, which is 30 and 60 cm/day, for Time=1, and 2, respectively. Individual nodal fluxes are readily available only in this file (they can be obtained also from “v.out”; Section 2.1.9; however, those are less accurate). The significance of the data is demonstrated in Figure 2.4, which shows fluxes through individual actual seepage points (Worksheet ‘Boundary’, Cells ‘C440, G444’ and Cells ‘C490, G496’).

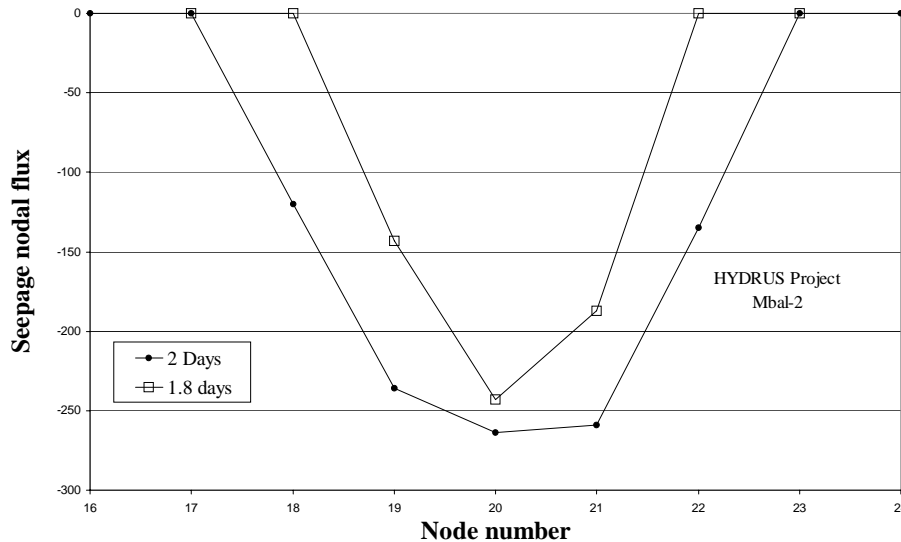


Figure 2.4: Individual nodal fluxes from ‘Boundary.out’ file

Column 6: Q refers to flow rate (cm^2/day). This is the flux (v) integrated over the nodal spacings (Figure 2.5). When integrating, the first and the last nodal fluxes (for $I=40$ and 44) will be multiplied by half the spacing only.

Note that if the flux at two neighbouring nodes (of an element) is different, it is common practice in finite element methods to assume a linear distribution of the flux along the element.

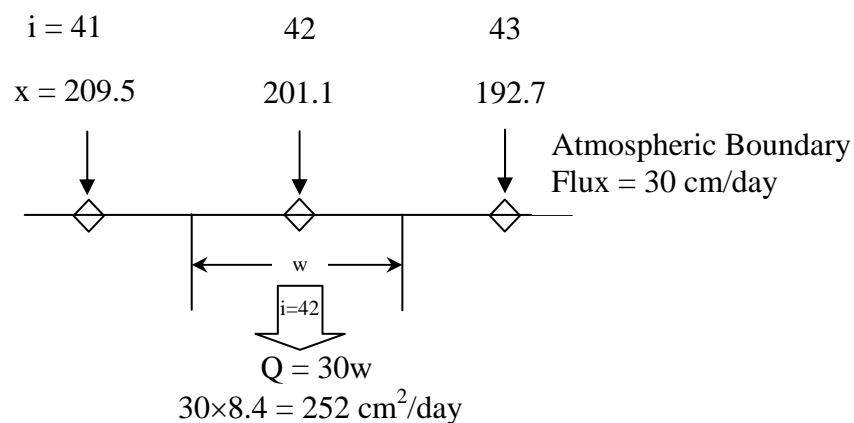


Figure 2.5: Definition of v and Q

Columns 7, 8, 9, and 10: h , th , $temp$, and $conc$, refer to pressure head (cm), water content, temperature, and solute concentration, respectively.

2. $v_Mean.out$ data at run time level (in Worksheet: v_Mean):

Column 1: Time (days) refers to run time. The smaller the time steps, the more records will be stored in this file. Time steps are controlled by the minimum and maximum time steps and by how quickly the solution converges (if the solution does not converge quickly, the time steps decrease and hence the number of records will increase).

Columns 2 and 4: rAtm and vAtm refer to potential and actual atmospheric flux (cm/day), respectively (equal in this problem).

Column 8: vSeep is the seepage flux across the boundary (cm²/day). Note that seepage starts at Time=1.6988 days (row 931 in spreadsheet). At time=1.8 days, $vSeep=574$ cm²/day. This value can be calculated from output file “Boundary.out” as follows:

Go to print time=1.8 day in Worksheet Boundary,
Sum Q in column G, cells G441+G442+G443=573 cm²/day

The mass balance may be calculated from this file as follows:

- Calculate Q_{atm} by multiplying the actual atmospheric flux by the width of the atmospheric BC (33.7 cm calculated in the previous section).
- Add Q_{atm} to Column (I, $vSeep$) to get the total In/out flow.
- This is demonstrated in column P, Worksheet v_Mean . The result is shown in Figure 2.6.

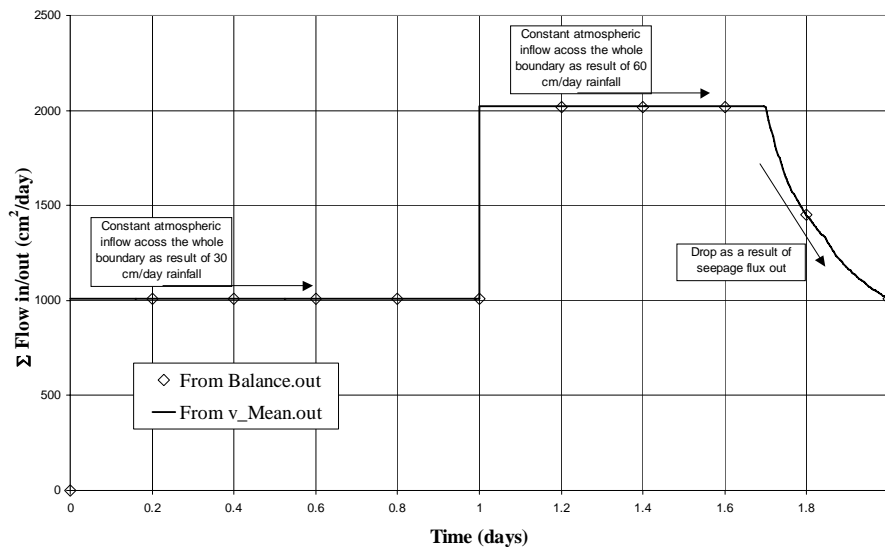


Figure 2.6: Sum of inflow and outflow rates

In the next section, the results shown in Figure 2.6 will be compared with those obtained from the “Balance.out” file at the print times.

The flow rates may be integrated over time to obtain cumulative fluxes identical to those found in output file “Cum_Q.out”. This is demonstrated in Columns M and N (Worksheet v_Mean) for cumulative atmospheric and seepage flows, respectively. The results are very close to the corresponding values reported in the “Cum_Q.out” output file (Columns E and I; Worksheet Cum_Q). The difference is less than 0.3%.

3. **Cum_Q.out** data at run time level (in Worksheet: Cum_Q):

Columns 2 and 4: CumQAP and CumQA refer to potential and actual cumulative atmospheric fluxes (cm^2), respectively (equal in this problem) across the entire atmospheric boundary.

Column 8: CumQS is the cumulative seepage flow volume (cm^2) across the entire seepage boundary.

4. **A_level.out** data at atmospheric time level (in Worksheet: A_level):

Columns 1: Time is atmospheric time, that is, any time where a variable atmospheric boundary condition has been specified. For our problem here this was at 1 and 2 days. We specified (under variable boundary conditions) that:
during time = 0-1 day, rainfall is 30 cm/day, and
during time = 1-2 day, rainfall is 60 cm/day.

Columns 2 and 4: CumQAP and CumQA refer to potential and actual cumulative atmospheric flow volumes (cm^2), respectively, across the whole atmospheric boundary. These numbers can also be found in Output file “Cum_Q” at times 1 and 2 days, respectively. They are calculated as follows:

$$\text{At time 1 } \underline{\text{CumQA}} = 30 \times 33.7 \times 1 = 1010 \text{ cm}^2$$

$$\text{At time 1 } \underline{\text{CumQA}} = 1010 + 60 \times 33.7 \times (2-1) = 3030 \text{ cm}^2$$

5. **Balance.out** data at Print time level (in Worksheet: Balance):

The mass balance can be calculated on a volume basis as follows:

Volume of water stored in soil prior to simulation, (Time=0) = 3990 cm^2 (Cell D13)

Volume of water stored in soil at end of simulation, (Time=2) = 6820 cm^2 (Cell D91)

$$\text{Change in water storage (gain)} = 6820 - 3990 = 2830 \text{ cm}^2$$

$$\text{Cumulative infiltration (rainfall into domain)} = -3040$$

$$\text{Cumulative seepage (out of domain)} = 202$$

$$\text{Mass balance (should be zero)} = 8$$

- For all print times, write the ‘InFlow’ values (Cells D14, D20, etc.) and the corresponding times in two neighbouring columns as shown in Cells (G12-H22). Insert the data set into Figure 2.6. Notice that you obtained the same results as from the calculations in file “v_Mean.out”, but only at the print times. To get a more continuous set of mass balance results, you can increase the print times up to 250 (or up to 5,000 if done outside the graphical interface; see Section 4.12).

2.2.3 Project “Mbal-3” (CD\HYDRUS Manual Projects\Sect-2): the domain of this problem is shown in Figure 2.7a. The upper 120 cm of the profile is sandy loam and the lower 20 cm is silt. Due to the configuration of the soil layers and boundary conditions, this system will promote lateral flow. The left-hand side of the upper boundary is an atmospheric BC, the lower boundary is a free drainage BC, and the right-hand vertical boundary is a seepage face. The soil profile is initially fully saturated, rainfall is simulated at a rate of 10 cm/day during the 0.2-day simulation. The number of print times was set to 100 to obtain a good resolution of nodal fluxes reported at print times in the “Boundary.out” output file.

In this example we will go through the mass balance calculations by considering subregions. Referring to Figure 2.7a, there are two subregions in this example (the domain is split vertically). The atmospheric boundary condition is restricted to subregion 1 (nodes 84-95), the free drainage BC goes through both regions (nodes 2-14 in subregion 1, and nodes 15-36 in subregion 2), while the seepage face is in subregion 2 (nodes 37-53).

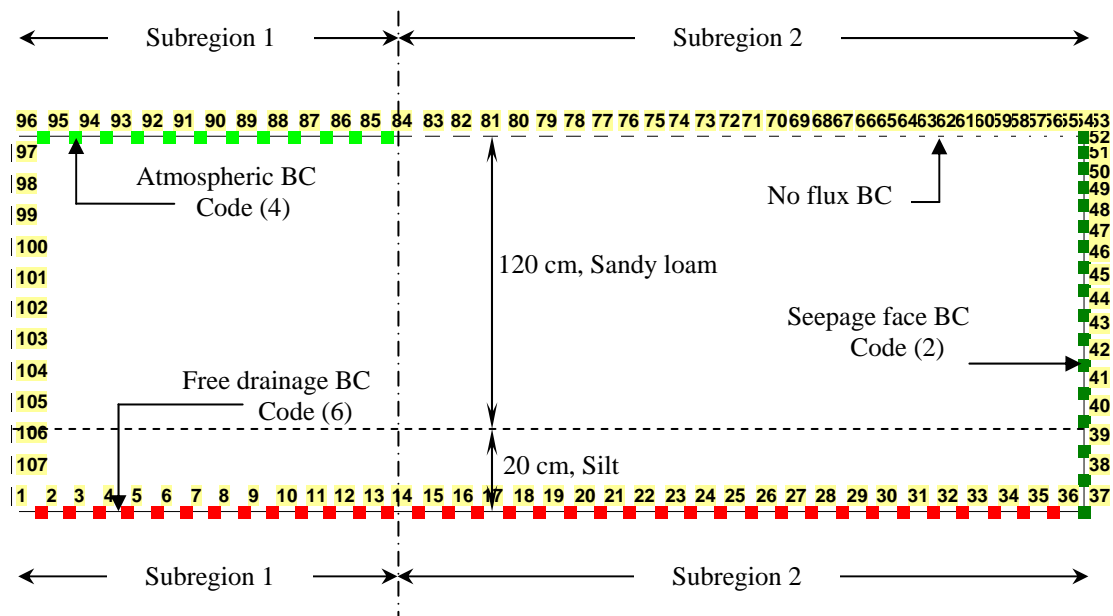


Figure 2.7a: Domain and BC for Project Mbal-3

We are interested in quantifying the amount of lateral flow from subregion 1 to subregion 2 (refer to the block diagram in Figure 2.7b). Let us study individual mass balances for each subregion at the end of the simulation (Time=0.2 day).

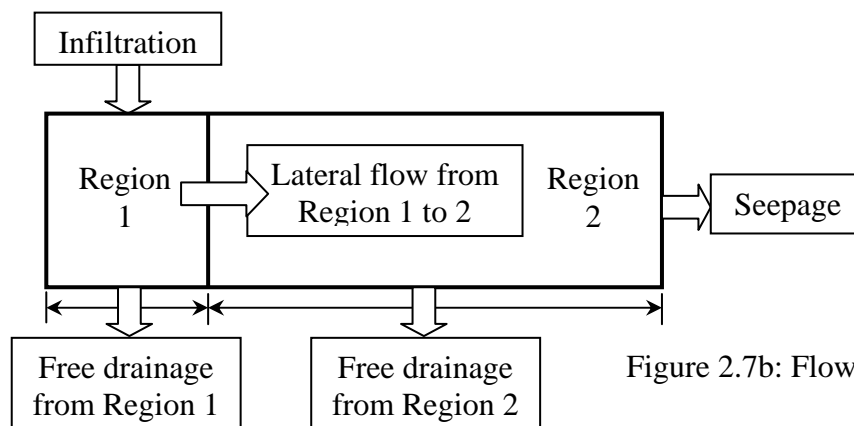


Figure 2.7b: Flow through regions

Cumulative fluxes are reported across entire boundaries (e.g., the flux for the free drainage boundary is reported across nodes 2-36). Hence, we must split this boundary in order to calculate how much water leaves subregion 1 (nodes 2-14) and how much leaves subregion 2 (nodes 15-36). This is achieved by manipulating the data reported in the “Boundary.out” output file.

Open file Mbal-3.xls in the CD (CD\HYDRUS Manual Projects\Sect-2).

The ‘Boundary.out’ output file is imported into Worksheet ‘Boundary’.

- For each print time,
 - Write the print time (1st print time is in Cell C10) in ‘Cell N6’.
 - Sum up Q’s for nodes $n = 2-14$ (G15-G27); this represent the flow rate from subregion 1 at the first Print Time. Write the result in ‘Cell O6’.
- Repeat 100 times to get data in Cells ‘N6-O105’ (writing a macro in Visual Basic helps).
- Integrate the flow rate over the print times to obtain volumes; if the print times are equal, simply multiply Q by the print time to get volume. If they are unequal, use an average. For example, to get the volume (V_x) at print time T_x :
 - $V_x = Q_x \times (T_{x-1} + T_x)/2$
 - The result is shown in Column ‘P’.
- Calculate cumulative volume as in Column ‘Q’.
- Repeat the same process by summing nodes 15-36 to get the cumulative volume from free drainage under subregion 2. The results are shown in (Cells S5-V105).
- Add the final numbers from both subregions; this should be equal to that reported in output file Cum_Q.out; referring to Worksheet ‘Boundary’:
 Cell Q105 + V105 = 161.82 + 298.32 = 460.14 \approx 460 (Sheet Cum_Q, Cell K142) OK.
- Mass balance for subregion 1 = Infiltration (water gained, negative sign) + Free boundary flux (sum of nodes 2-14, water lost) + change in storage (note that water drained from the soil is a gain to the water balance, and thus has a negative sign; calculated as the difference in Volume at time 0.2 day and the initial Volume, Worksheet Balance) + Lateral flow from subregions ‘1’ to ‘2’ = 0 (only lateral flow is unknown)
- The amount of lateral flow that resulted in a zero-mass balance is added as ‘flow in’ to subregion ‘2’; the water balance for subregion ‘2’ is calculated in the same way; the lateral out flow from subregion ‘2’ should be equal to the cumulative seepage reported in output file ‘Cum_Q.out, Cell I142’.

A sample calculation is shown in Worksheet ‘Volume-based mass balance’, and is also shown in Table 2.2.

Cumulative Boundary Flow from 'Cum_Q.out' output file found in Worksheet Cum_Q

Cumulative Free Boundary Flow Cell K-142	460
Cumulative Seepage Flow Cell I-142	906
Cumulative Actual Atmospheric Flow Cell E-142	-257

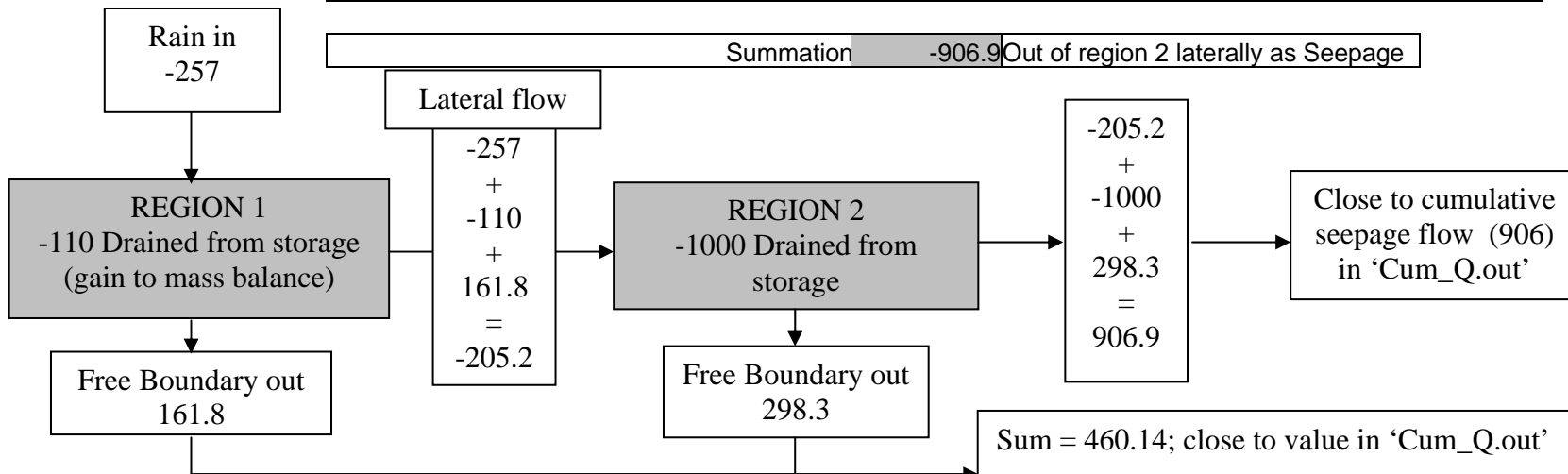
Table 2.2: Volume-based mass balance (from spreadsheet of same name in Workbook Mbal-3.xls)

Mass Balance Region 1

			Spreadsheet	Cell
Atmospheric Boundary, Rain =	-257	gained	Cum_Q	E142
Free Boundary flow out under Region 1 =	161.8	lost	Boundary	Q105
Change in Soil-water storage (drained) =	-110	gained	Balance	O19
Summation	-205.2		Out of region 1 laterally into Region 2	

Mass Balance Region 2

			Spreadsheet	Cell
Input from Region 1 =	-205.18	gained	This sheet	E13
Free Boundary flow out under Region 1 =	298.3	lost	Boundary	V105
Change in Soil-water storage (drained) =	-1000	gained	Balance	P19
Summation	-906.9		Out of region 2 laterally as Seepage	



Compare the flow rates during the simulation from output files 'v_Mean.out' and 'Balance.out'.

- Flow rates at run times are found in output file 'v_Mean.out'. In the case of atmospheric BCs, fluxes are reported (cm/day). Hence, we sum them up over the entire boundary to obtain the total flow rate (cm²/day). This is achieved by multiplying the flux by the width of the atmospheric BC, which is 130 cm in this case. The calculations are shown in Worksheet 'Flow rates, Columns C and D'.
- Add 'Column D' (Worksheet 'Flow rates') to 'Columns I and K' (Worksheet 'v_Mean'); find the result in 'Column H' (Worksheet 'Flow rates'). This will result in total inflow/outflow rates during the simulation.
- Plot the result versus time as shown in Figure 2.8.

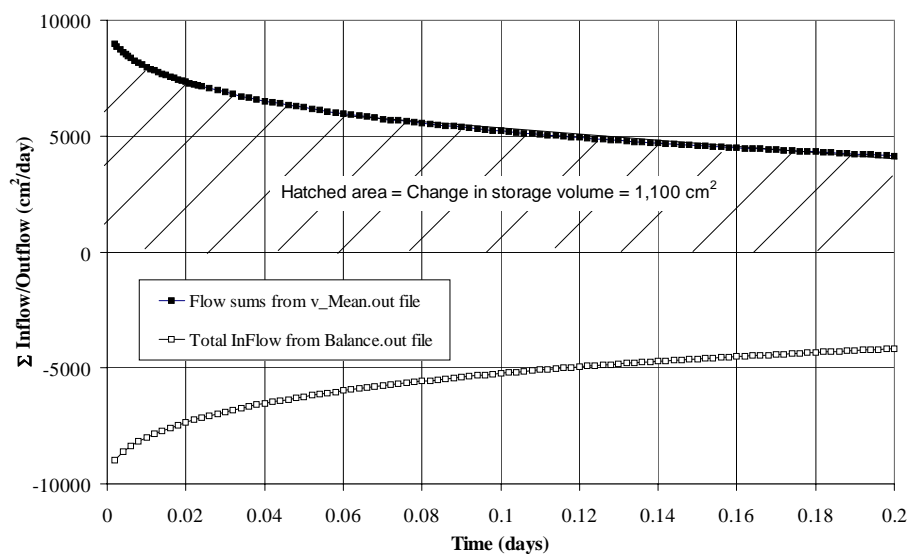


Figure 2.8 : Total inflow/outflow from 'Balance.out' and 'v_Mean.out' files

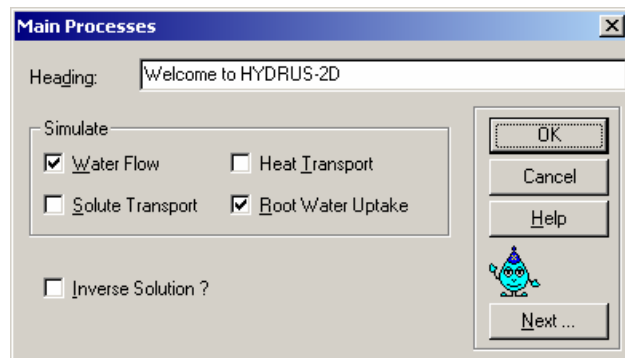
- The same data may be obtained from output file 'Balance.out'.
- Go to Worksheet 'Balance'.
- For each print time,
 - Write the print time (1st print time is found in Cell B11) in 'Cell G3'
 - Write Inflow (1st Inflow is found in Cell D14) in 'Cell I3'.
- Repeat 100 times to get data in Cells 'G3-I103'.
- Plot inflow data versus time as shown in Figure 2.8; you will see that they are similar to those obtained from the 'v_Mean.out' file.

If we integrate flow rates over time we obtain cumulative fluxes (or volumes). The area under the curve shown in Figure 2.8 represents the change in storage during the simulation (1100 cm²).

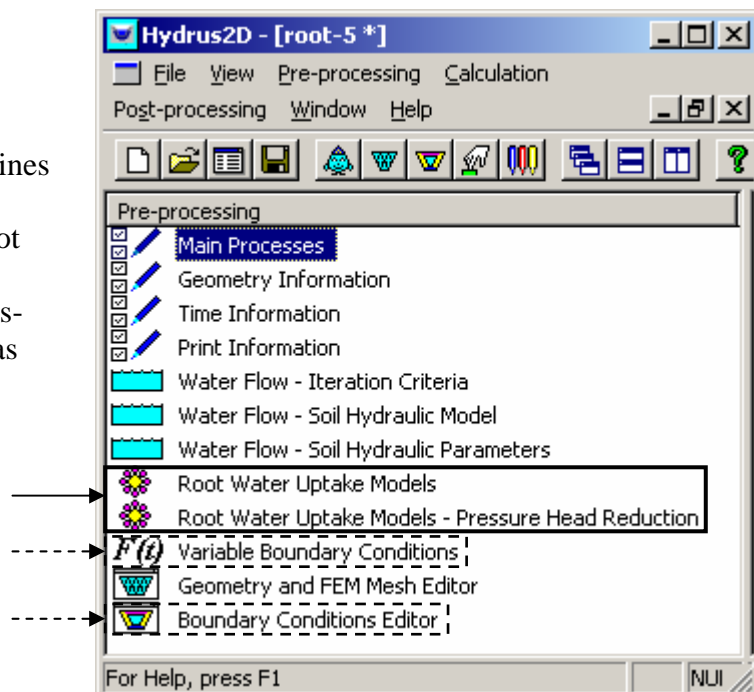
3. ROOT WATER UPTAKE

3.1 Relevant Windows in HYDRUS

When root water uptake (plant transpiration) is modelled, the relevant box in “Main Processes” should be checked.



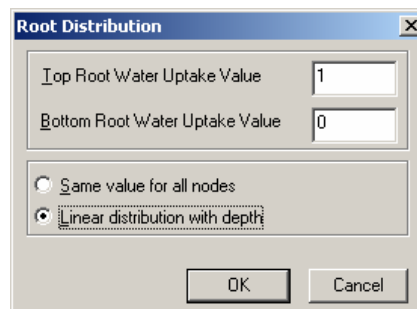
As a result, two new input lines are added under “Pre-Processing”. They are: “Root Water Uptake Models” and “Root Water Uptake Models-Pressure Head Reduction” as shown here.



Other related windows where input is required are:

- “Variable Boundary Conditions”, enter ‘potential transpiration’. Described in Boundary conditions, Section 1.1.10-9; Atmospheric.
- “Boundary Condition Editor”
 - Either:
 - “Condition/Root Distribution” (🌱)
 - Select nodes
 - Set Value
 - Water Uptake Distribution; enter appropriate value

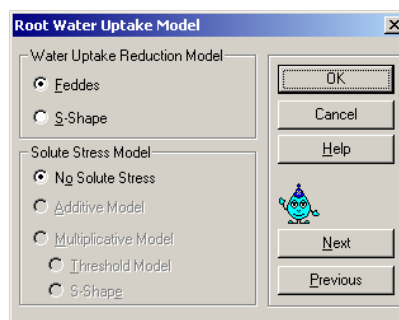
- Select a set of nodes, and choose one of the options as shown in this window. When selecting ‘Same value for all nodes’, only the ‘Top Root Uptake Value’ is needed. Refer to Appendix III.2 for details.



- *Alternatively*, you can enter appropriate parameters under “Option/Parameters for Root Distribution”; see Appendix III.3 for details.

Root Water Uptake Model:

The water uptake reduction model defines the manner in which transpiration is reduced below the potential rate when the soil is no longer capable of supplying the amount of water demanded by the plant under the prevailing weather conditions. There are two alternative reduction models: one by the Feddes et al. (1978), further referred to as the Feddes model, and one by van Genuchten (1987), further referred to as the S-shaped model. The former is more commonly used.



Feddes' Model Parameters

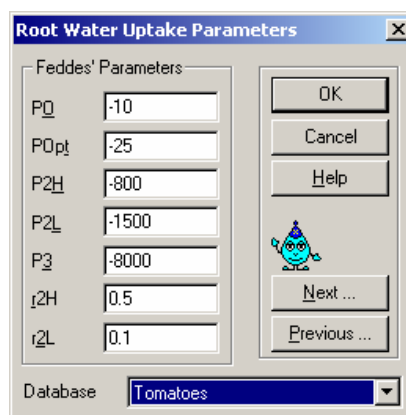
The Feddes model assigns plant transpiration rates according to the soil's pressure head.

Feddes' model parameters are shown in this window. The meaning of those parameters is graphically demonstrated in Appendix III.1.

P_0 : Value of the pressure head below which roots start to extract water from the soil.

P_{opt} : Value of the pressure head below which roots extract water at the maximum possible rate (potential transpiration).

P_{2H} : Value of the limiting pressure head below which roots no longer extract water at the maximum rate (assuming a potential transpiration rate of r_{2H}).



P2L: As above, but for a potential transpiration rate of r2L.

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

A database of suggested values for different plants is provided based on studies by Wesseling (1991) and Taylor and Ashcroft (1972).

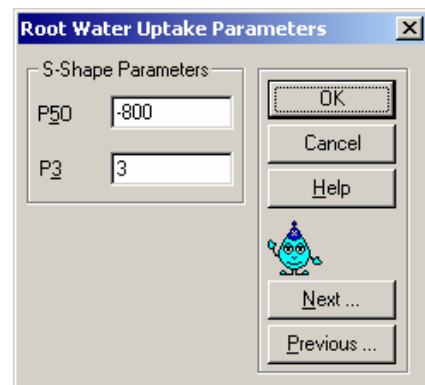
r2H: Potential transpiration rate (L/T) (currently set at 0.5 cm/day).

r2L: Potential transpiration rate (L/T) (currently set at 0.1 cm/day).

The above 2 input parameters permit one to make the variable P2 a function of the potential transpiration rate (P2 presumably decreases at higher transpiration rates). HYDRUS currently implements a linear interpolation scheme for this purpose (given in the HYDRUS Technical Manual).

S-Shaped Model Parameters

van Genuchten (1987) proposed an S-shaped function to describe root water uptake reduction, which is applicable to water stress as well as salinity stress. Two input parameters are required in this model: P3, which is an experimental constant, and P50, the pressure head at which the actual uptake rate is half the potential rate. More details about the S-shaped function are found in Appendix III.1.



3.2 Root water uptake simulations

Projects Root-1 to Root-8 (**CD\HYDRUS Manual Projects\Sect-3**) demonstrate the effects of various root water uptake parameters on root zone fluxes and soil water pressure heads. The projects simulate root water uptake from a freely draining, 1-m deep silty-loam soil profile.

The initial pressure head for all projects is hydrostatic equilibrium with the zero pressure head at the freely draining bottom boundary (initially set equal to zero).

Projects Root-1, 2 and 3: Effect of range of optimal root water uptake (difference between P2H and P2L).

The Projects assume a uniform root distribution down to a depth of 50 cm (no roots below 50 cm), and a potential transpiration rate of 0.2 cm/day. Note that in the case of a uniform root distribution, the absolute value for the water uptake distribution does not really matter; any number that you enter will yield the same result since the root distribution values are internally integrated and normalized by HYDRUS .

Project Root-4: As above but the root distribution gradually decreases from unity at the surface to zero at a depth of 50 cm.

Results:

Projects Root- 1 and 2: Figure 3.1 shows that the actual flux falls below the potential rate at a later time when P2L is higher. In other words, root water uptake stays at the potential rate longer.

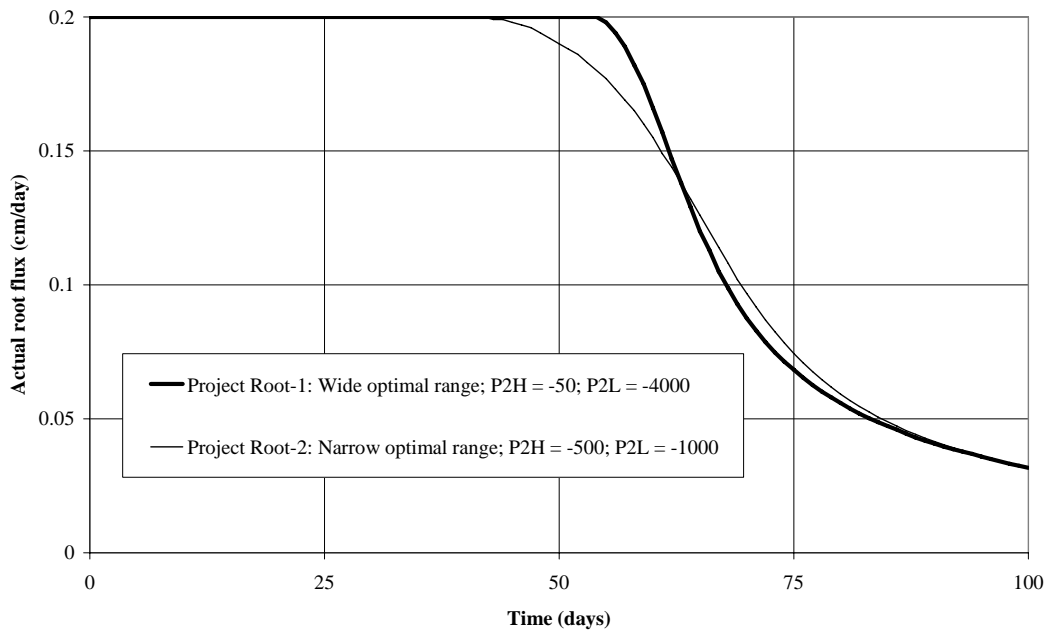


Figure 3.1: Effect of P2H and P2L on calculated actual root fluxes

Projects Root- 1 and 2: Figure 3.2 shows that since actual root uptake is reduced at a later stage (high P2L), the pressure heads in Simulation Root-1 are lower than those in Simulation Root-2.

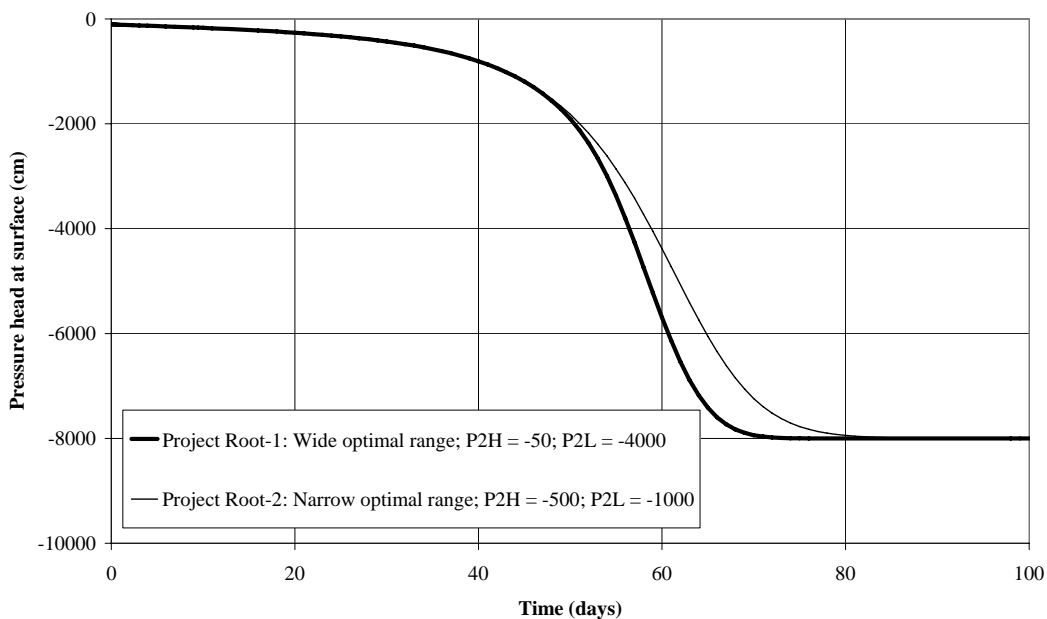


Figure 3.2: Effect of P2H and P2L on calculated soil water pressure heads

Project Root-3: Figure 3.3 shows that toward the end of the relatively long simulation, the flux fluctuates due to upward migration of moisture caused by the gradient between the dry soil within the root zone (at depth=50 cm) and the soil below it. This process is the result of the selected precision for calculations. Uptake rates at larger time are very small (<0.001 cm/d) and thus below the water content precision tolerance ($=0.001$). Oscillations can be eliminated by decreasing the water content tolerance to 0.0001.

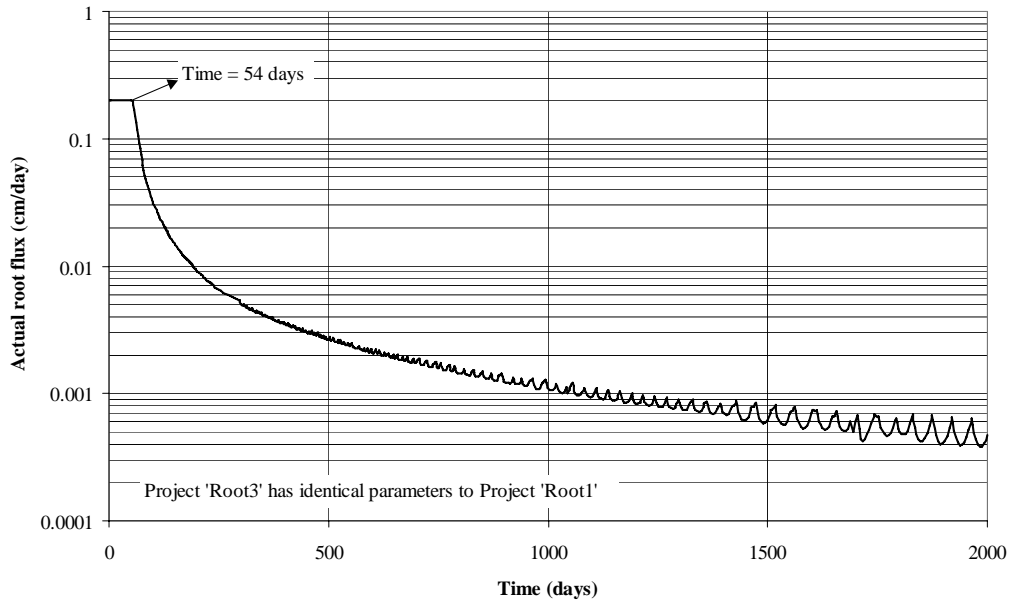


Figure 3.3: Root fluxes with abrupt change in root distribution (Root-3)

Figure 3.4 shows that the soil within the root zone eventually reaches a pressure head equal to P_3 (-8,000 cm in this example). Notice that at a depth of 52 cm (just outside the root zone), the pressure head does not reach that value.

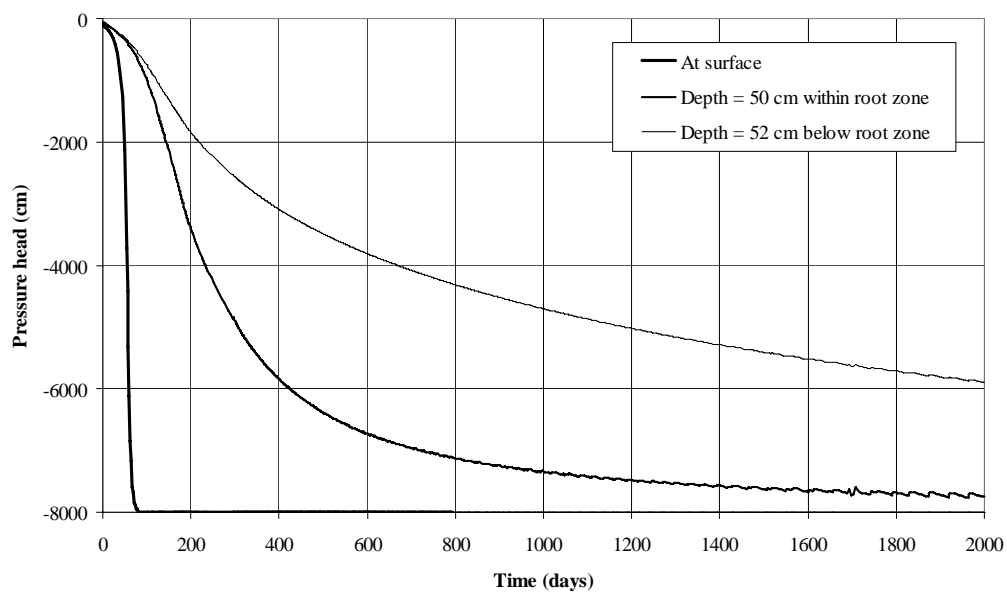


Figure 3.4: Calculated pressure heads at various depths (Root-3)

Projects Root- 1 and 4: in Project Root-4, the root distribution gradually decreases from unity at the surface to zero at a depth of 50 cm. Referring to Figure 3.5, notice that for a uniform root distribution at different depths, there is a lag in the pressure head change; however, the trends are identical.

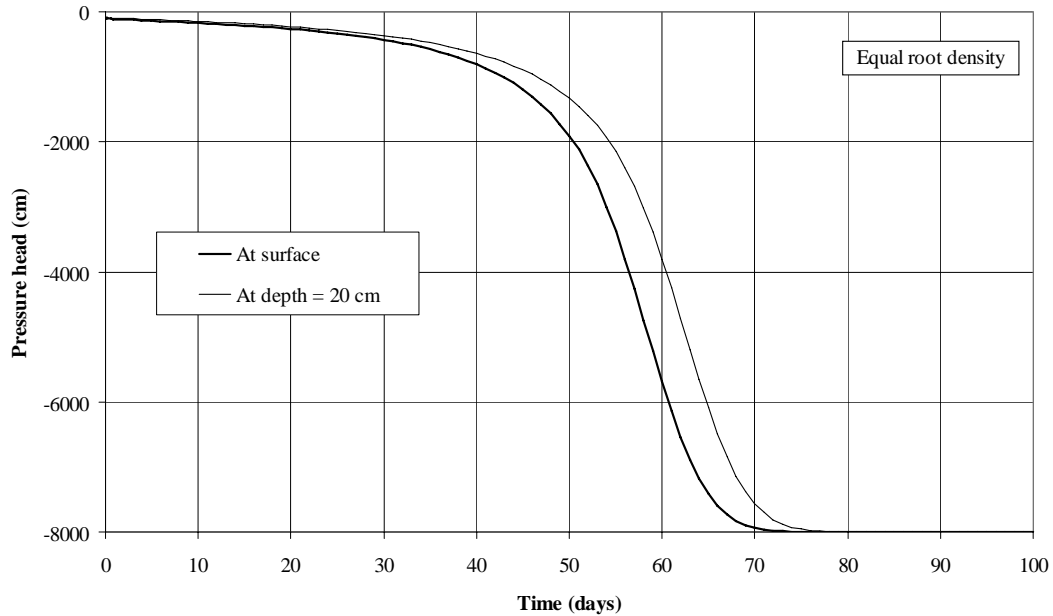


Figure 3.5: Calculated pressure heads for uniform root distribution (Root-1)

Figure 3.6 shows that when the root densities are different at two locations in the root zone, the pressure heads will evolve very differently in time.

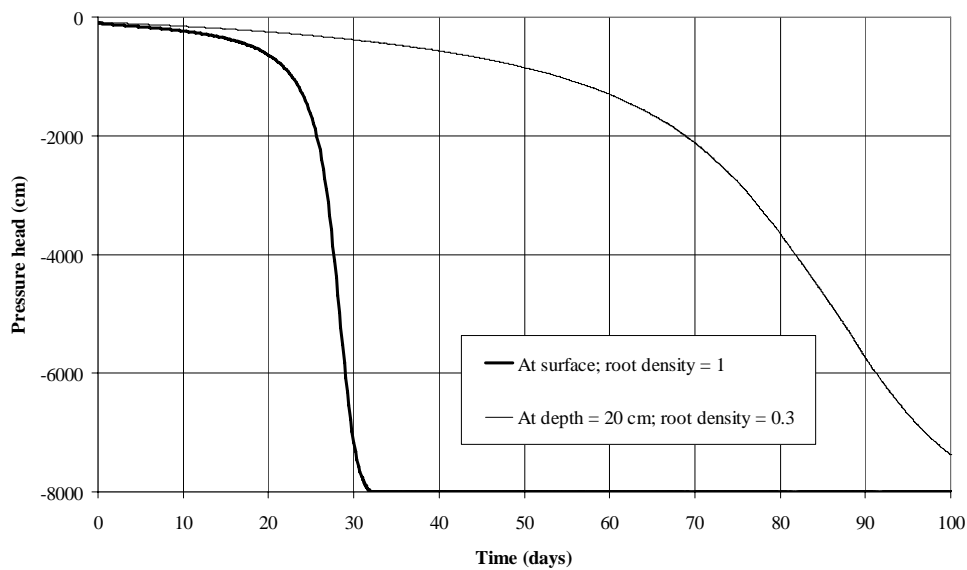


Figure 3.6: Calculated pressure heads for non-uniform root distribution

Projects Root-5: This simulation is identical to Root-1, except that the potential transpiration rate is doubled to 0.4 cm/day. Figure 3.7 shows that increasing the potential transpiration rate causes the pressure heads to drop much earlier; however, the trends are similar in both cases.

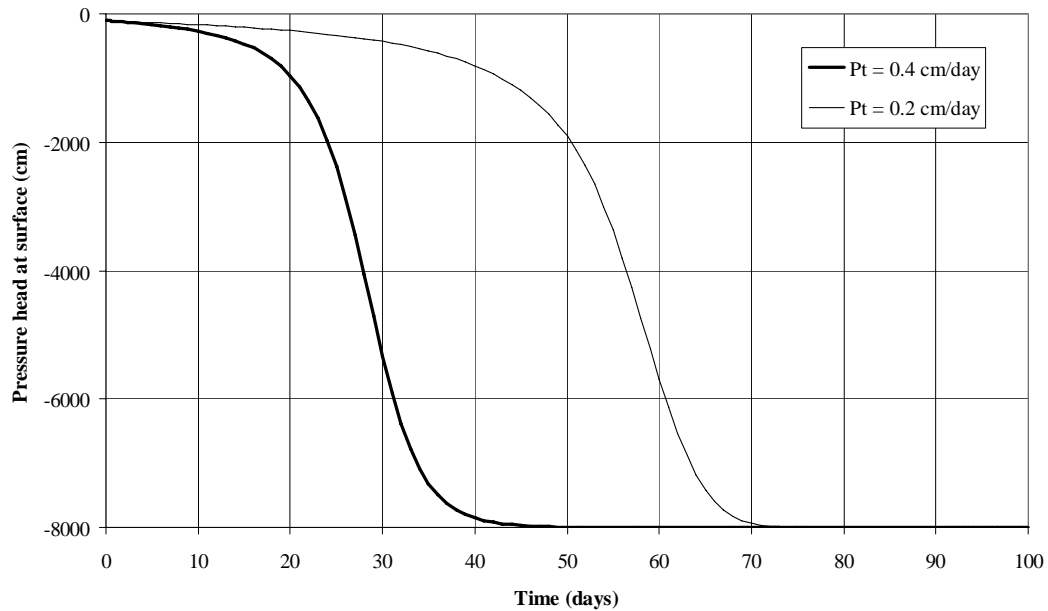


Figure 3.7: Effect of potential transpiration rate (Pt) on calculated pressure heads

Figure 3.8 shows the effect of increasing the potential transpiration rate on cumulative root extraction.

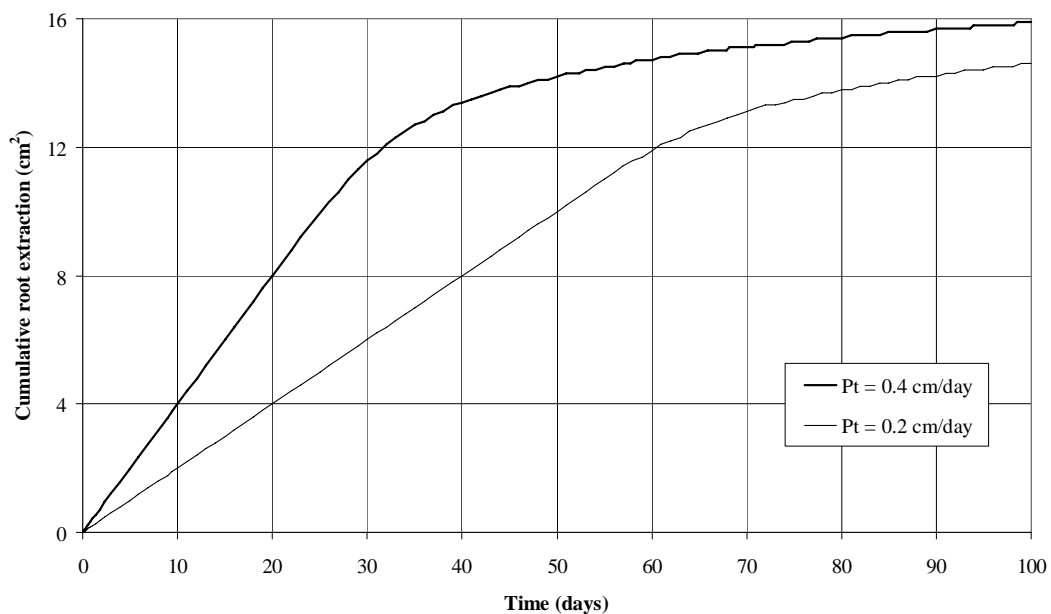


Figure 3.8: Effect of the potential transpiration rate on cumulative root extraction

Project Root-6: Effect of initial conditions

In this simulation, the initial pressure head of the soil is higher than P_0 (i.e., nearly at saturation). The lower boundary is sealed (not drained). Since the pressure head is higher than P_0 everywhere in the domain, it will stay that way and transpiration will never start. This problem will be addressed in Projects Root- 7 and 8.

Project Root-7: The surface nodes (only the top 2 nodes) are set to a pressure equal to P_0 . The results are shown in Figure 3.9.

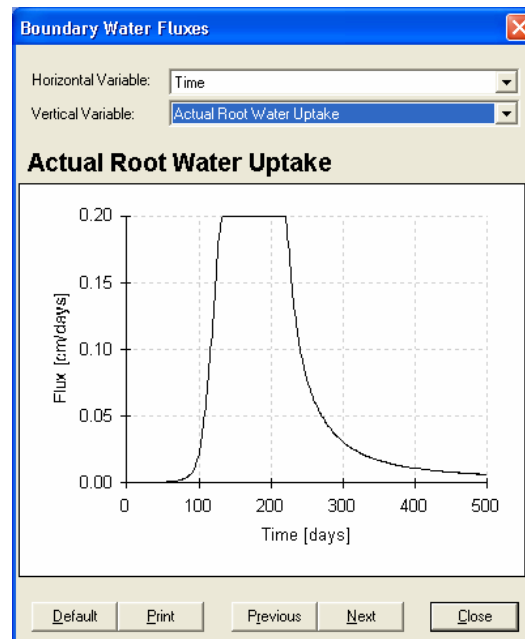


Figure 3.9: Actual root uptake for Project Root-7

Project Root-8: Introduce surface evaporation for a short duration of time. The time variable BC records are increased to 2. The first record represents an evaporation period of 0.1 day at an evaporation rate of 0.1 cm/day. This will decrease the surface pressure head to a value below P_0 , which in turn induces the initiation of transpiration.

4. EXAMPLE APPLICATIONS

4.1 Fluctuating Stream Level

In this section, we model a stream with a fluctuating water level. When the water level in the stream is rising, a variable head BC is suitable. The specified pressure head is assigned to the lowest point and hydrostatic equilibrium is assumed for the points above it. For a falling stream stage, this boundary does not represent what actually happens since a seepage face likely will develop. HYDRUS at present is not capable of modelling such a problem in one project because of the changing nature of the boundary representing the stream. Therefore, simulations must be carried out in two stages as follows:

1. Project “Ris-St”: simulate the rising water level using a variable head boundary representing the stream.
2. Project “Fall-St”: simulate the water table drawdown that results from the lowered stream level. The initial conditions for this project are imported from the last time level in Project “Ris-St”.

All related Projects are found on the CD (..HYDRUS Manual Projects(Sect-4)).

Project “Ris-St”

The conceptual model for this problem is shown in Figure 4.1. The soil used is a sandy loam. The flow domain is initially in hydrostatic equilibrium with the water table that is located at the lower base of the domain. The stream rises in three stages as shown in Figure 4.1.

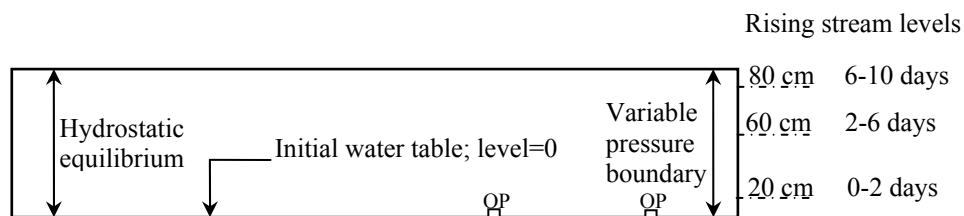


Figure 4.1: Conceptual model for Project “Ris-St”

Figure 4.2 shows the pressure head response at 50 cm and 150 cm away from the stream obtained at two observation points (OP in Figure 4.1).

Figure 4.2: Response to an increase in the stream level

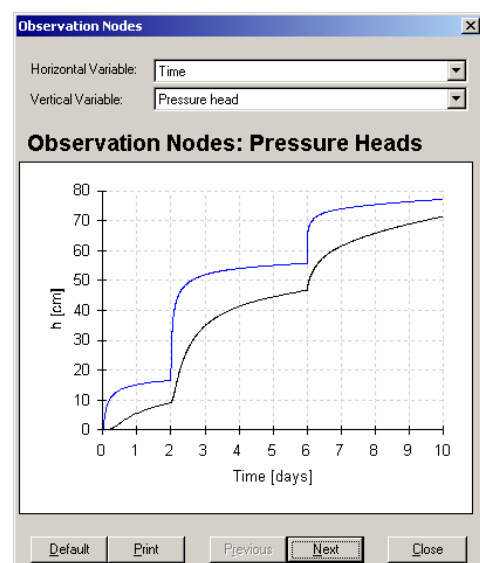


Figure 4.3 shows the pressure head distribution across the entire domain at Time=10 days. This distribution will be used as the initial condition for the next project.

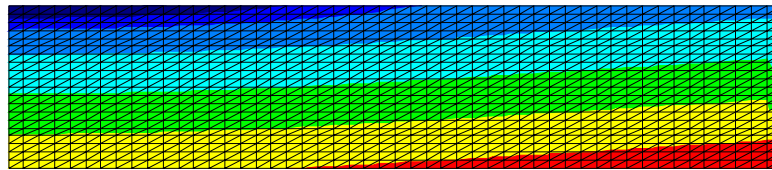
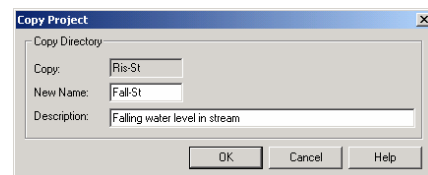


Figure 4.3: Pressure head distribution; Time=10 days

Project Manager: Select Project “Ris-St”

Copy, and change details as shown here. This will provide us with an identical copy of the project. We will only change the initial conditions and the boundary conditions representing the stream.



Project: “Fall-St”

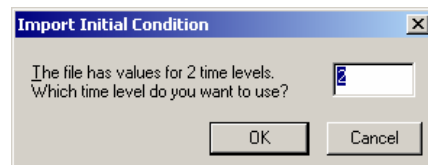
Boundary Condition Editor ()

Sidebar Menu: Water Flow Buttons/Seepage face: select the vertical points representing the stream down to a depth of 40 cm.

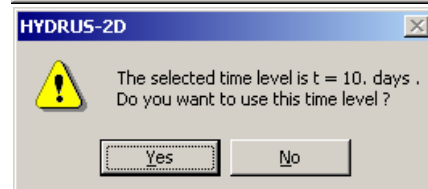
Condition/Initial Condition/Import Pressure

Directory path.....\Ris-St\h.out\Open

Accept the default value 2 (as shown here).



Accept the default value $t=10$ days (as shown here).



The initial pressure head distribution should be identical to that shown in Figure 4.3.

Figure 4.4 shows the pressure head response to the falling-stream stage at the two observation nodes. Note that the initial time of the simulation is day 10, which is the final time of the previous simulation; this is set in “Time Information/Initial Time”.

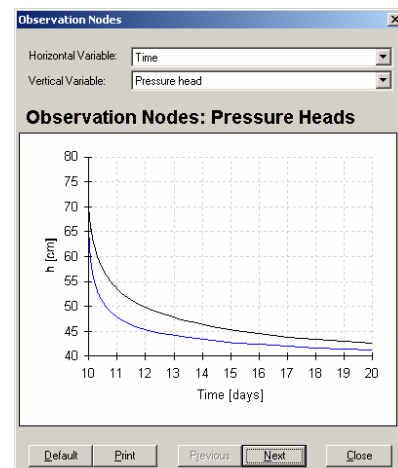


Figure 4.4: Response to drop in stream level

4.2 Modelling Evaporation: Effect of Discretization

The spacing of the elements near the soil surface can have a significant impact on the results of evaporation simulations. The effect is related to movement of the drying front through the soil profile and development of potentially very low (negative) pressure heads (and gradients) near the soil surface. All related Projects are found on the CD (**..\HYDRUS Manual Projects\Sect-4**).

4.2.1 Advance of Drying Boundary

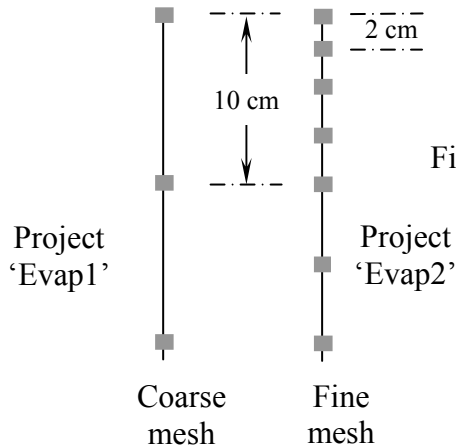


Figure 4.5: Grid spacing near the soil surface

In projects “Evap1” and “Evap2”, we track the movement of the drying front for two contrasting meshes, coarse and fine, respectively (see Figure 4.5).

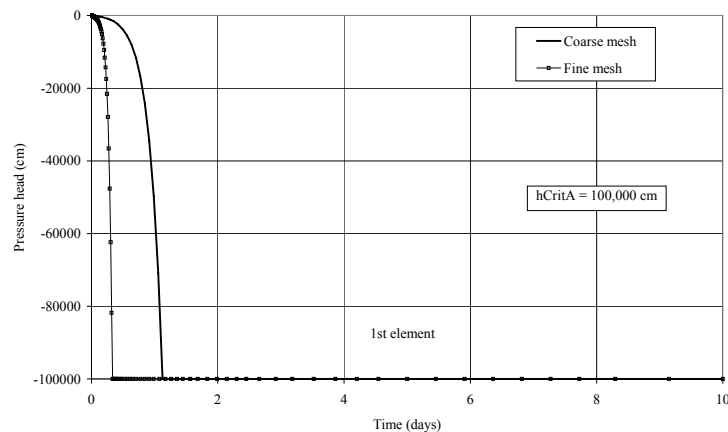


Figure 4.6: Surface pressure heads during evaporation as calculated using fine and coarse meshes

Figure 4.6 shows the pressure head time series for the two meshes. Due to the shallower depth of the 1st element in the fine mesh, h_{CritA} was reached at an earlier time; evaporation stage 2 hence commenced earlier. h_{CritA} is the minimum allowed pressure head at the soil surface (set at a very low $-100,000$ cm in this example). For more details about h_{CritA} , refer to Appendix II.

Figure 4.7 shows pressure head data at deeper points in the profile. For the fine mesh we see that the drying front has advanced to the second node (at 2 cm). However, the front has not progressed as much for the coarse mesh because of the larger element size (10 cm). For the fine mesh, at 10 cm depth, pressure heads are still very low since the drying front moves through the soil profile on an element-by-element basis depending upon the pressure gradients between them.

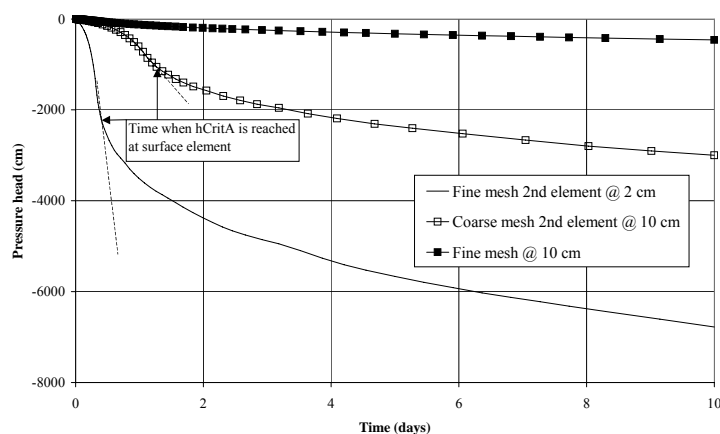


Figure 4.7: Pressure heads at two depths during evaporation as calculated using fine and coarse meshes

This phenomenon indicates that modellers should investigate the suitability of the mesh when simulating evaporation. In the following section, we will demonstrate the effect of mesh size on predicted evaporative fluxes.

4.2.2 Mesh Density and Evaporative Fluxes

We demonstrate the effect of mesh density by conducting four simulations having rectangular grids of varying density near the soil surface. We achieve this by changing the total number of elements as well as the node density at the soil surface (Density at Top, in Space Discretization). Mesh details for the four simulations are listed in Table 4.1.

Figure 4.8 shows the significant effect of spatial discretization on the predicted evaporative fluxes. However, these changes tend to become less significant at some level of mesh density. Apparently the coarser mesh (“Cmesh”) is not an option; this mesh is far too coarse and overestimates the flux. The predictions tend to converge to some value as the mesh is made finer. Of course, a mesh that is very fine may take too long to run; see Table 4.1 for run times. We conclude that for this set of simulations the best mesh is “Vfmesh” since it produces results very close to those obtained with mesh “VVFmesh” in less than one-tenth of the run time.

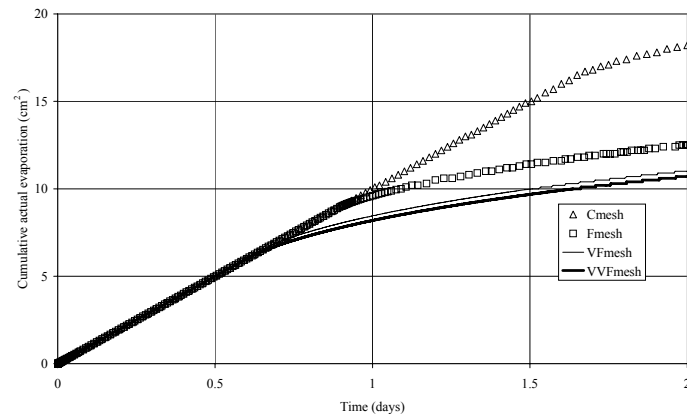


Figure 4.8: Effect of mesh density on evaporative fluxes

Table 4.1: Run details of rectangular 100-cm deep grid used to simulate evaporation

Project name	Number of elements	Density at top	Depth of surface element (cm)	Run time (Seconds)
Cmesh	20	1	5	1
Fmesh	100	1	1	20
VFmesh	100	0.1	0.2	43
VVFmesh	200	0.05	0.05	652

Concluding remarks:

1. Mesh density has a significant effect on predictions of evaporative fluxes since it impacts how the drying front advances.
2. When modelling evaporation, it is worthwhile to investigate the appropriateness of the mesh beforehand. In addition, a value of h_{CritA} that best suits the soil modelled must be identified (refer to Appendix II).

4.3 Capillary Barrier

The principle of a capillary barrier is having a fine-textured soil layer situated on top of a coarse-textured soil layer. The goal is to minimize water leaching into the lower layer. Capillary barriers are widely used in mining applications and landfills where a soil cover (the fine material) retains the water and loses it to evaporation and/or transpiration with a minimum leaching-component into the underlying waste material (such as mine waste rock or tailings). Project “Mesh9” is found on the CD (**..HYDRUS Manual Projects\Sect-4**).

In Project Mesh9, we simulate a 1-dimensional freely draining soil profile that comprises 2 layers; an 80-cm fine layer that overlays a 20-cm coarse layer. The optimum mesh parameters established in Project ‘Mesh8e’ (of Section 6.4) are adopted here to simulate this practical application.

The results shown in the three figures below were obtained for two observation points located close to the interface separating the two layers (see Project Mesh9). Figure 4.9 shows the pressure head data for the two layers. The wetting front arrives just after 0.4 day. Both pressure heads increase at comparable rates.

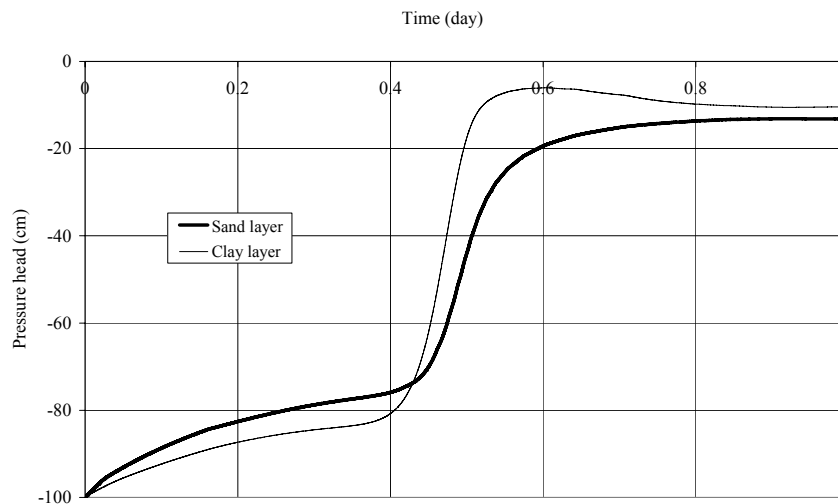


Figure 4.9. Pressure head data for sand and clay layers

Figure 4.10 shows the manner in which the water content varies in the two layers. Notice that there is a clear lag in response, with the sand keeping a much lower water content after the pressure head in the clay layer had already increased significantly.

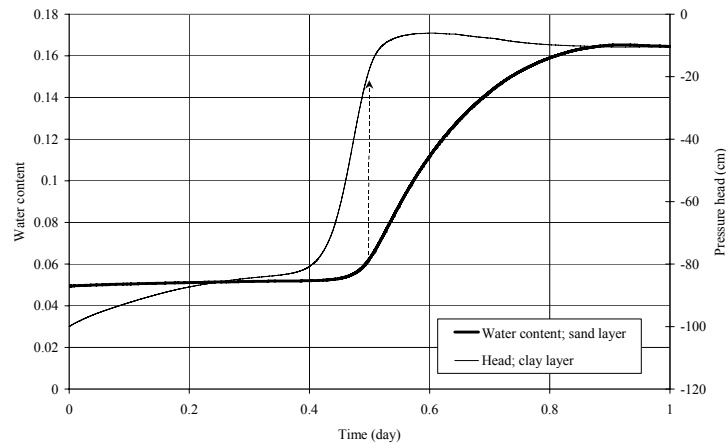


Figure 4.10. Water content and head data for sand and clay layers, respectively

In Figure 4.11, the water content is represented in relative terms as degree of saturation. The simulation shows that the clay reached 98% saturation at $t=0.5$ d while the sand is still at 3%, eventually reaching only about 30% saturation.

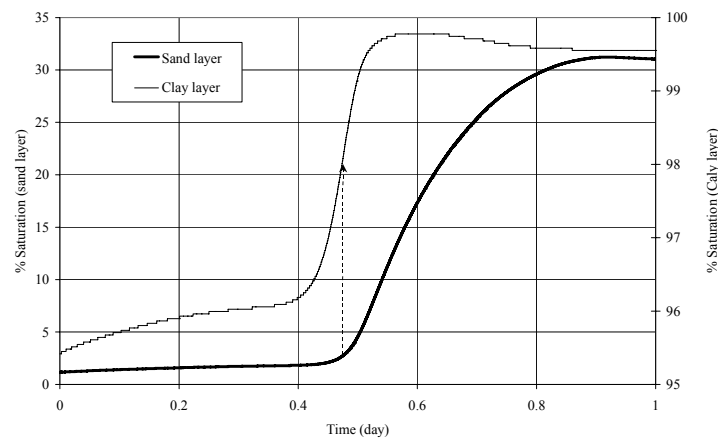


Figure 4.11. Relative water content (% saturation) for sand and clay layers

Figure 4.12 shows that the cumulative free drainage flow, which may represent a hazardous leachate, has dramatically dropped from 42.6 cm^2 (for Project Mesh-8e of Section 6.4) to only 3.77 cm^2 .

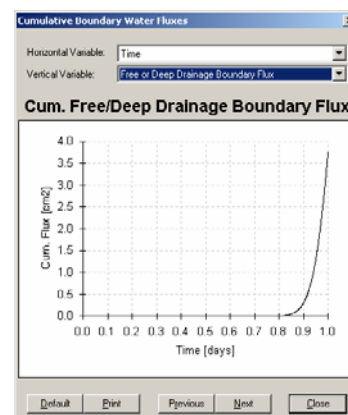


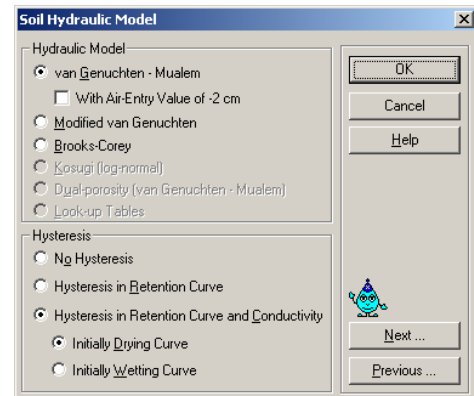
Figure 4.12. Cumulative drainage boundary flux for a layered system

4.4 Modelling Hysteresis

Hysteresis refers to the non-unique relationship between pressure head and water content in the soil water retention curve. A drying cycle requires more energy (a higher negative pressure head) to arrive at the same water content compared to a wetting cycle. That is, it is easier to wet the soil than to dry it to the same water content. It is possible to incorporate hysteresis with HYDRUS only when using the van Genuchten-Mualem soil hydraulic model.

Hysteresis can be extended also to the conductivity function.

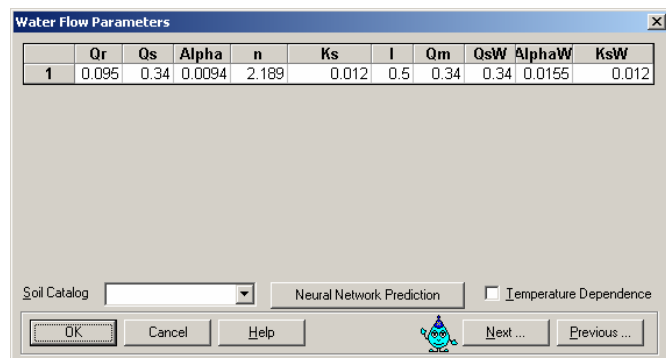
One should specify where the simulation starts, i.e. whether the initial water contents and pressure heads are on a drying or on a wetting curve.



Drying curve parameters

Q_r : residual water content
 Q_s : saturated water content
 Alpha, n, K_s , I, and Q_m .

Note that Q_m in this window refers to a drying parameter. The meaning of Q_m is explained in the modified van Genuchten model (see Appendix I).



Wetting curve parameters

Q_sW : saturated water content
 AlphaW: parameter Alpha for wetting
 K_sW : saturated hydraulic conductivity
 Q_m for wetting is calculated by HYDRUS such that

$$(Q_s - Q_r)_{\text{drying}} / (Q_s - Q_r)_{\text{wetting}} = (Q_m - Q_r)_{\text{drying}} / (Q_m - Q_r)_{\text{wetting}}$$

Q_r and n are usually equal for drying and wetting, whereas Alpha for wetting is larger than that for drying (Šimůnek et al., 1999). When no data are available for AlphaW, a good initial guess is $\text{AlphaW} = 2 \times \text{Alpha}$.

In the following section, we will briefly investigate the effect of hysteresis by comparing results obtained from three simulation series: (1) Using drying data and no hysteresis, (2) Using wetting data and no hysteresis, and (3) Using wetting and drying

data with hysteresis. All related Projects are found on the CD (**..\HYDRUS Manual Projects\Sect-4**).

The parameters used in the following simulations are listed in Table 4.2. The resulting drying and wetting water retention curves are shown in Figure 4.13.

Table 4.2: Input parameters for hysteresis simulations

Q_r	Q_s	Alpha	n	K_s	Q_m	Q_sW	AlphaW	K_sW
0.095	0.34	0.0094	2.189	0.012	0.34	0.34	0.0155	0.012

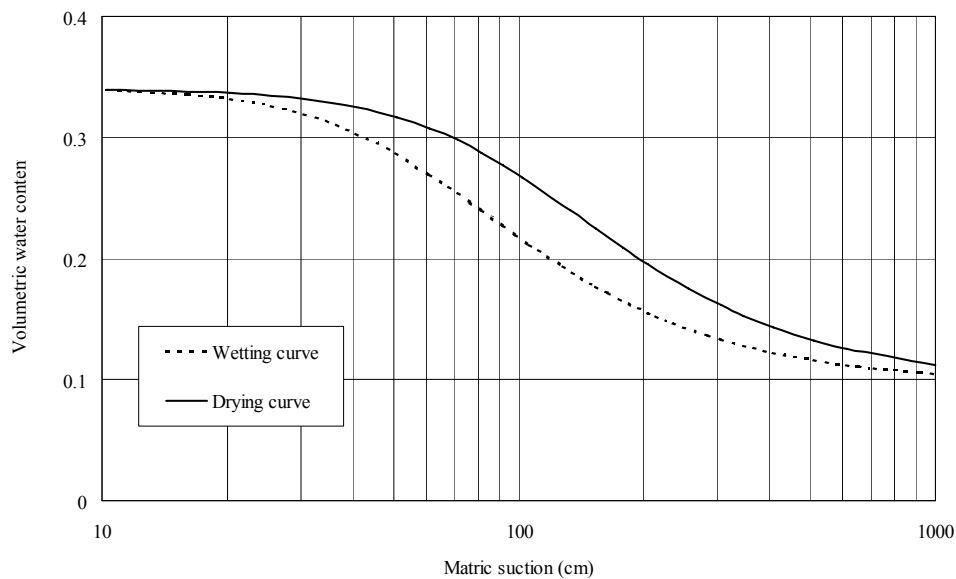


Figure 4.13: Drying and wetting water retention curves

Details of the simulations are outlined in Section 5.2.1. We model infiltration into an initially dry soil sample; a drying cycle is then initiated when the surface flux is terminated; another wetting cycle is subsequently simulated when the flux is reintroduced to the soil surface.

Refer to Project ‘Cyc-1’ on the CD (**..\HYDRUS Manual Projects\Sect-4**). Figure 4.14 shows that some numerically induced fluctuations were encountered during the second wetting cycle. Altering the iteration criteria solves the problem (see Table 4.3).

Table 4.3: Iteration criteria details for hysteresis simulations

	Cyc-1	Cyc-2
Water content tolerance	0.0001	0.00001
Pressure head tolerance	0.1	0.01
Lower limit of tension interval	0.01	0.0001
Upper limit of tension interval	200	1000

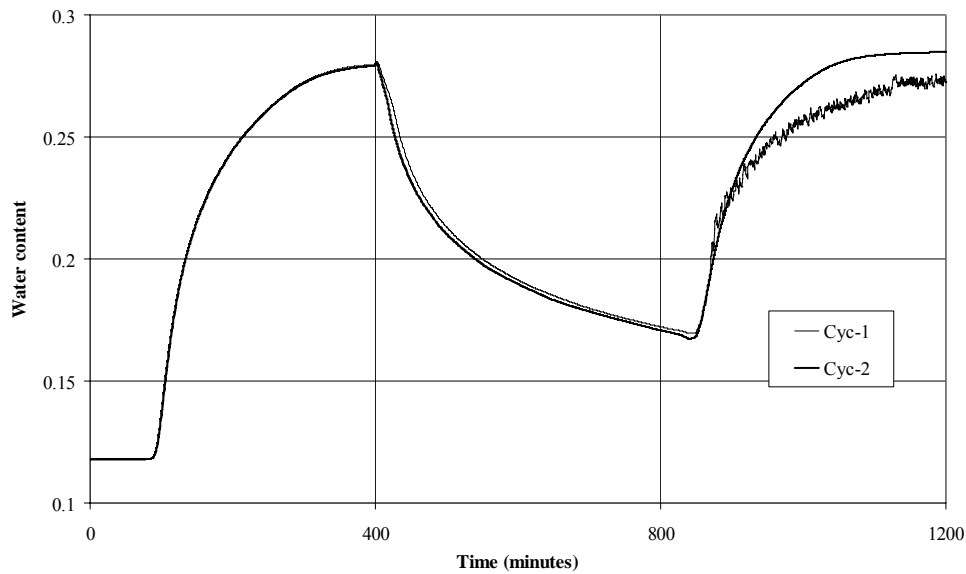


Figure 4.14: Comparison of simulations 'Cyc-1' and 'Cyc-2'

For projects 'Drying' and 'Wetting', we use drying and wetting water retention parameters without hysteresis, respectively. Figure 4.15 compares the resulting pressure heads for the two cases. Pressure head predictions obtained from project 'Cyc-2', which incorporates hysteresis, are compared with the previous two runs (see Figure 4.15). Note how the pressure head prediction with hysteresis moves from the wetting to the drying cycle, and then again to the wetting cycle.

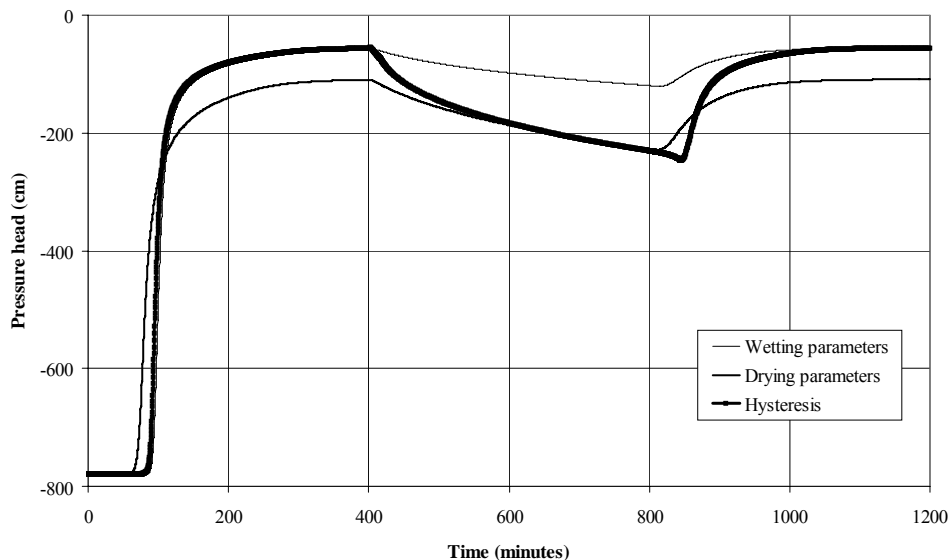


Figure 4.15: Comparison of pressure head predictions with wetting, drying, and hysteresis parameters

Water content predictions are compared in Figure 4.16. When hysteresis is modelled, the predicted water content is mostly higher during the wetting cycle and lower during the drying cycle, as compared to the other two scenarios.

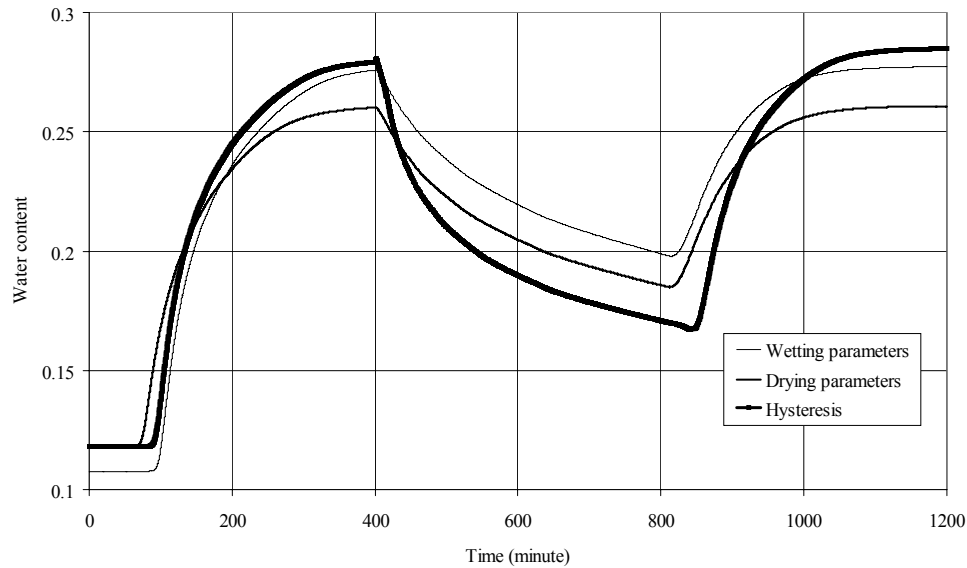


Figure 4.16: Comparison of water content predictions with wetting, drying, and hysteresis parameters

Figure 4.17 shows that the three approaches gave different predictions for soil water storage. The storage term is obtained from the output file 'Balance.out' (the water volume stored in the soil is represented by the variable "Volume" in this output file). Notice that consideration of hysteresis may have implications in the design of soil covers.

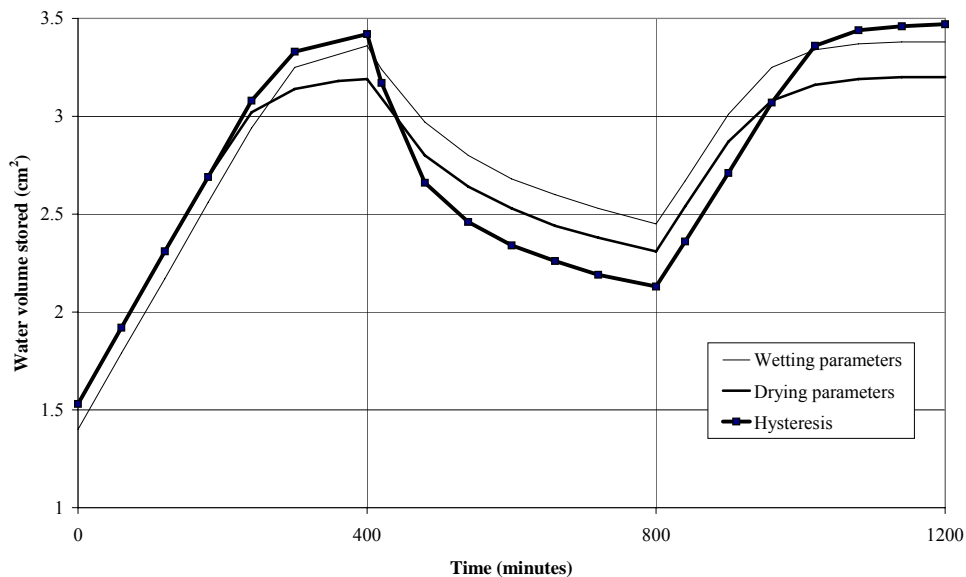


Figure 4.17: Comparison of water storage predictions with wetting, drying, and hysteresis parameters

4.5 Estimating Flow Rates Inside a Domain

In HYDRUS, fluxes at the run time levels are only reported along external boundaries. Fluxes (or flow velocities) may be integrated (summed) along external boundaries to obtain rates from parts of a boundary (done outside HYDRUS; see Section 2.2.3). This information is not readily available for locations inside the domain, but still can be calculated. Nodal flow velocities for the entire domain are only available at print times. They can either be obtained in text format for the whole domain (in file “v.out”; see Section 2.1.9), or for individual sections throughout the domain within the graphical interface. In both cases, to obtain flow rates across an internal line, the integration should be handled externally (e.g., in a spreadsheet). In this exercise, the velocity profiles across internal line are obtained from the Post-Processor Graphical Interface. Turning the ‘Snap to Grid’ on facilitates the selection of the end points. Numerical integration is performed in a spreadsheet using the trapezoidal rule.

The project series (Pipe-) simulates flow through a hypothetical set-up where two compartments are connected by a 10-cm wide conduit. Compartment ‘A’ has a constant head boundary on its surface and compartment ‘B’ has a seepage face boundary on its side. We wish to estimate the velocity distribution across various sections along the conduit length, estimate the flow rate through the conduit, and compare it with the inflow and outflow rates through the external boundaries.

The advantage with this project is that we have a priori knowledge of the flow rate; mass balance stipulates that at any time, the flux through the pipe must be equal to the fluxes across the constant head boundary inflow rate, and the seepage face boundary outflow rate (all three fluxes are equal). We will demonstrate the effect of mesh discretization on the results. All related Projects are found on the CD (..\HYDRUS Manual Projects\Sect-4).

Project Pipe-1

Details of this project are shown in Figure 4.18. The default mesh that HYDRUS produces is accepted.

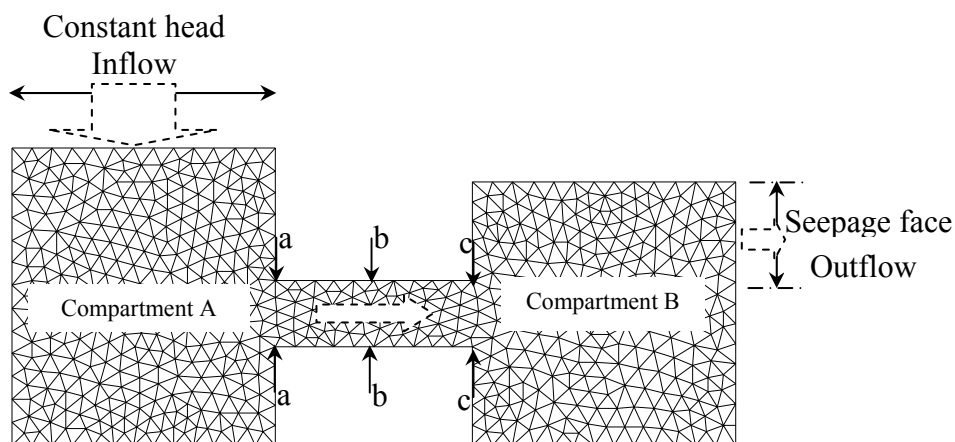


Figure 4.18: Conceptual model and FE mesh for Project Pipe-1

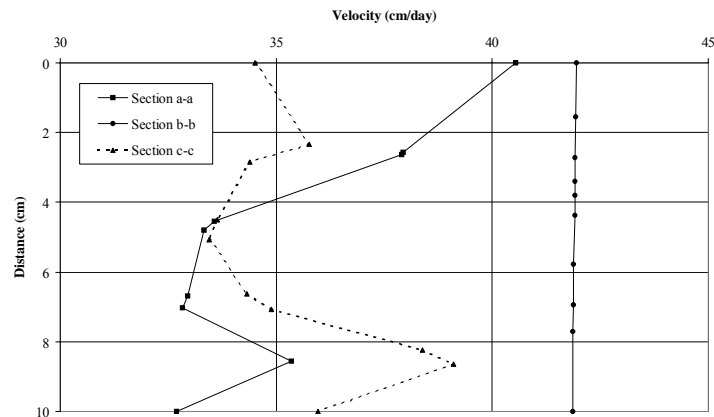


Figure 4.19: Velocity profiles for Project Pipe-1

The velocity distributions along three cross-sections in the conduit are shown in Figure 4.19 (refer to Figure 4.18 for the section locations). The velocity is uniform along the middle section. Integrating the velocities along this cross-section resulted in a flow rate identical to that calculated from the constant head boundary and the seepage face.

However, Table 4.4 shows that the flow rates obtained from the other two cross-sections have produced significant errors. Notice that the velocity profiles along sections a-a and c-c are highly non-linear especially near the top. This problem will be addressed in the following projects.

Table 4.4: Results from for various projects

	Project Pipe-1 Section b-b	Project Pipe-1 Sections a-a & c-c	Project Pipe-2	Project Pipe-3	Pipe-3SS Steady state analysis
Flow rates at internal x-sections cm^2/day	418.978	355; 352	890.27	940.25	940.25
Error %	0.00048	15.2; 15.7	6.94	1.72	1.72
External boundary flow rate cm^2/day	418.98		956.7		

Projects Pipe-2

In this project, the seepage face extends through the entire RHS boundary (see Figure 4.20) and the conduit is shorter. These changes will increase the nonlinearity in the velocity profile in the conduit. We will obtain velocity profiles for cross-section a-a only (highest non-linearity) and try to get the best possible results by manipulating the mesh.

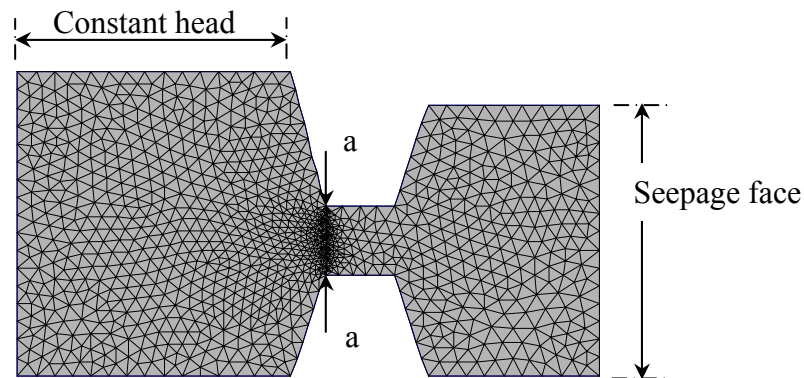


Figure 4.20: Finite element mesh for Projects Pipe-2

Let us refine the mesh in this project along cross-section 'a-a' (see Figure 4.20). This should result in a more refined velocity profile, and hence improve numerical integration.

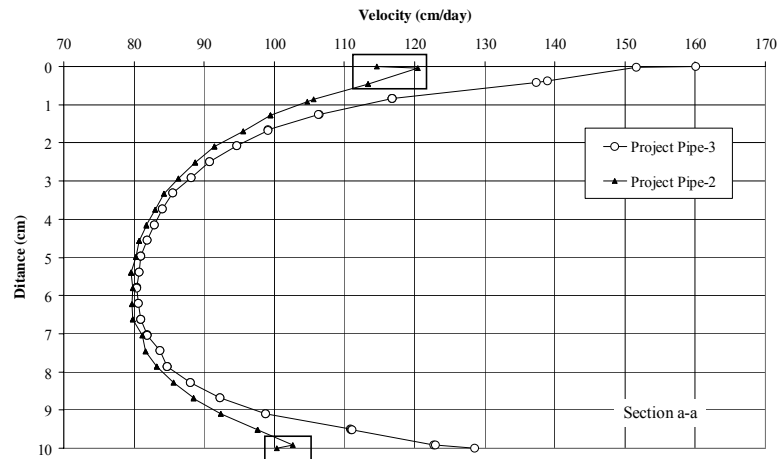


Figure 4.21: Velocity profiles at section a-a for Projects Pipe- 2 and 3

Figure 4.21 shows that the velocity profile is smoother; notice that we now have 28 points defining the profile as compared to only 10 in the previous example. Table 4.4 shows that error has dropped to 6.94%, but it is still significant.

Projects Pipe3

In this project, the mesh representing the entire area of the conduit is refined (see Figure 4.22). Figure 4.21 shows now a very smooth velocity profile was obtained. Table 4.4 indicates that the error has dropped to only 1.72%.

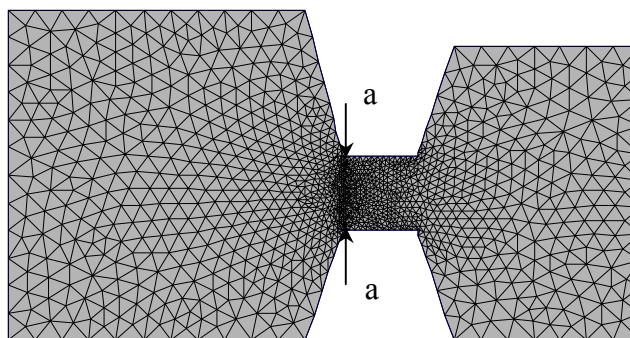


Figure 4.22: Finite element mesh for Projects Pipe-3

Figure 4.23 shows velocity contours obtained for this simulation; Notice the very high velocities at the corners. The simulation was repeated to obtain a steady-state solution; the results were identical (see Project Pipe-3SS; (CD\HYDRUS Manual Projects\Sect-4)).

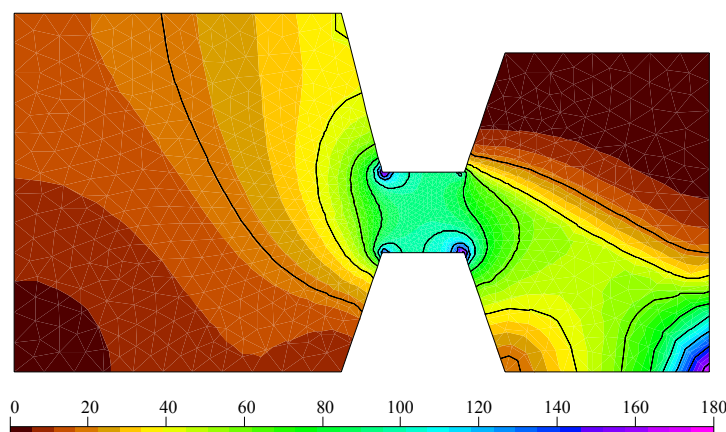


Figure 4.23: Velocity contours at steady state

Comments:

- It is possible to obtain flow rates across internal cross-sections inside the domain. Obtain the velocity profile and integrate outside HYDRUS (e.g., using a spreadsheet).
- You are likely to obtain better results if the velocity distribution across the section is either constant or varies in a linear manner.
- It is very important to investigate the existence of velocity nonlinearity across a section and discretize the mesh accordingly. That is, the higher the non-linearity, the finer the mesh should be.
- Note that this procedure is possible only at print times. Therefore, when deciding on print times, one should take into account the temporal nonlinearity that may exist in some problems (e.g. high early drainage rate in the case of a coarse soil).
- For external boundaries, the velocities reported in the Graphical interface are slightly different (less accurate) than those reported in the “Boundary.out” file.
- Velocities are secondary variables calculated by applying Darcy’s law to the pressure head (primary variable) field. Their precision is lower than the solution of the flow field itself (as obtained with the Richards equation).

4.6 Effect of Mesh Spatial Discretization and Tolerance Limits

Projects ‘cmbal’ and ‘fMbal1’, and ‘fMBal2’ demonstrate the effects of mesh density and tolerance limits on the simulation results, and subsequent impacts on the mass balance. The Projects are found on the CD (..\HYDRUS Manual Projects\Sect-4).

The three projects simulate a rectangular domain with an atmospheric BC at the surface, a free drainage BC at the bottom, and a seepage face on part of the RHS boundary. Infiltration is simulated as follows:

0-2 days 100 cm/day
 2-4 days -
 4-6 days 25 cm/day

The first two projects ‘cmbal’ and ‘fMbal1’ differ in terms of their mesh size, with the former having a relatively coarsely-spaced mesh. Project ‘fMbal2’ has a mesh size similar to Project ‘fMbal1’ but with looser tolerances. Details are listed in Table 4.5.

Table 4.5: Summary of mesh, mass balance, run time, and tolerance details

Project name	Element size (cm)	Number of elements	Tolerance		Run time (sec.)	Mass balance	
			WC	Pres.		V	%
cMBal	50×50	120	0.0001*	0.1*	19	-232	0.046
fMBal1	10×10	3000	0.0001*	0.1*	1947	-64	0.013
fMBal2	10×10	3000	0.01 [#]	0.5	1459	-20,200	4.134

*Default values

[#]High tolerance; not recommended.

Table 4.5 shows that the much finer grid produced a better mass balance. However, notice that this required a 100-fold increase in run time. Loosening the tolerances resulted in a significant error in the mass balance. The run time decreased by 25 %. Figure 4.24 compares cumulative free drainage fluxes for the three simulations.

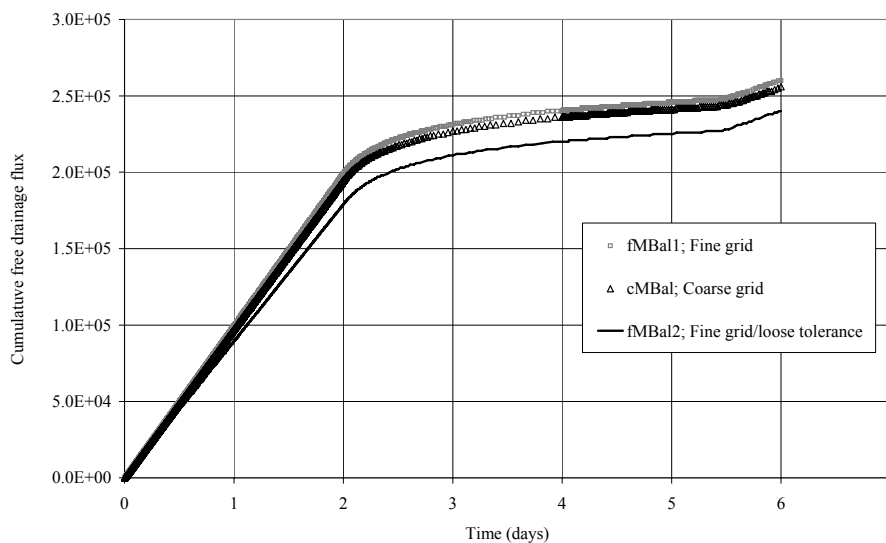


Figure 4.24. Comparison of cumulative free drainage flux for three projects

4.7 Simulating Surface Runoff

HYDRUS does not evaluate surface runoff. When precipitation exceeds the infiltration capacity of the soil, it instantaneously removes excess water from the soil surface. However, it is possible to mimic surface runoff by using an imaginary surface layer having a very high hydraulic conductivity and a saturated water content approaching unity. The excess water that cannot infiltrate layer 2 will move laterally and ultimately drain through a seepage face located at the end of this imaginary layer. Note that this technique should not be used if evaporation is simulated. The Project is found on the CD (..HYDRUS Manual Projects\Sect-4).

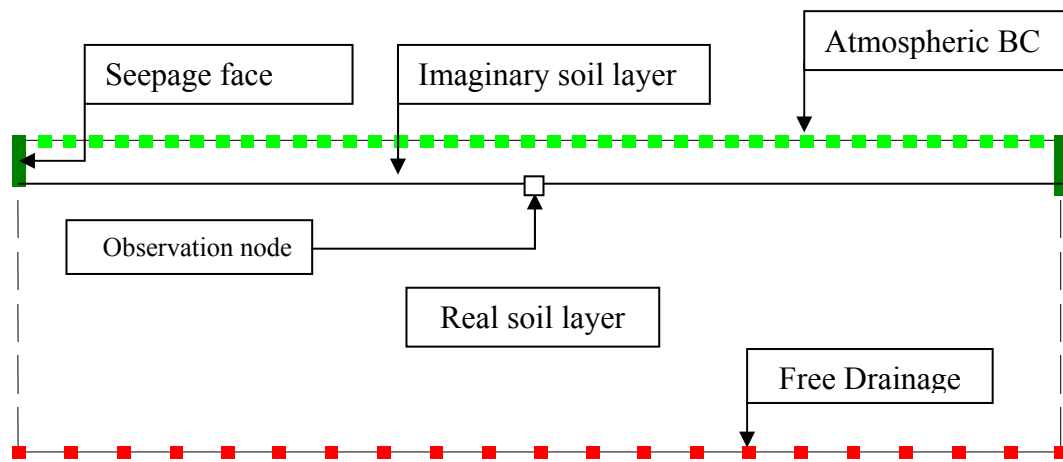


Figure 4.25: Boundary conditions for Project 'Runoff'

Project 'Runoff'

The scenario shown in Figure 4.25 is modelled in this project. Figure 4.26 shows the cumulative flow for the atmospheric and free drainage boundary, the difference is the runoff represented by the seepage component. The observation node shows that the water ponds to a height of about 2.2 cm on top of the real soil layer (results not shown here; see values for the observation node in the HYDRUS project).

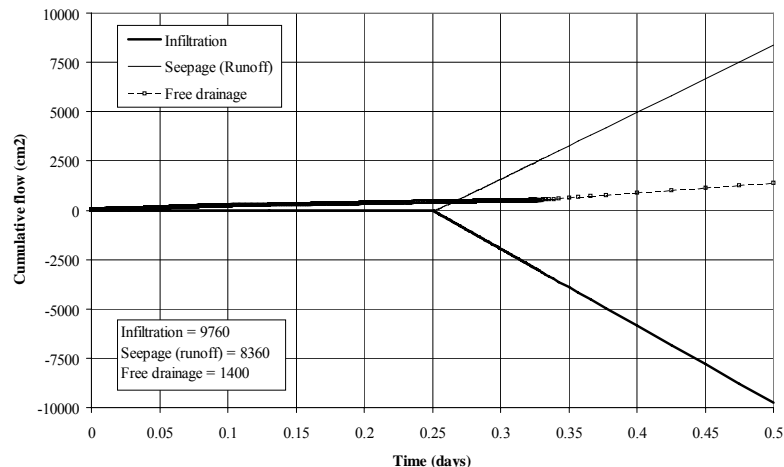


Figure 4.26: Cumulative flow components for project 'Runoff'

4.8: Local Anisotropy

In this hypothetical example we demonstrate the use of the ‘Local Anisotropy’ feature in HYDRUS. Refer to Section 5.3.12 for the theoretical background on ‘Implementation of hydraulic conductivity anisotropy’ (p. 55; HYDRUS Technical Manual).

We use the FE mesh constructed in Section 1.1.9.2 (Example 2, Project ‘Grid-2’), which represents a 15% sloping hill. We assume that a fault exists along the slope that acts as a preferential path for water draining along the slope. The hydraulic conductivity along the preferential path is equal to $50K_s$ (K_s is the saturated hydraulic conductivity elsewhere in the domain). Details of the domain, the boundary conditions, and the fault location are shown in Figure 4.27. We simulate precipitation for 1 day; the initial pressure head is -1 cm for the entire domain.

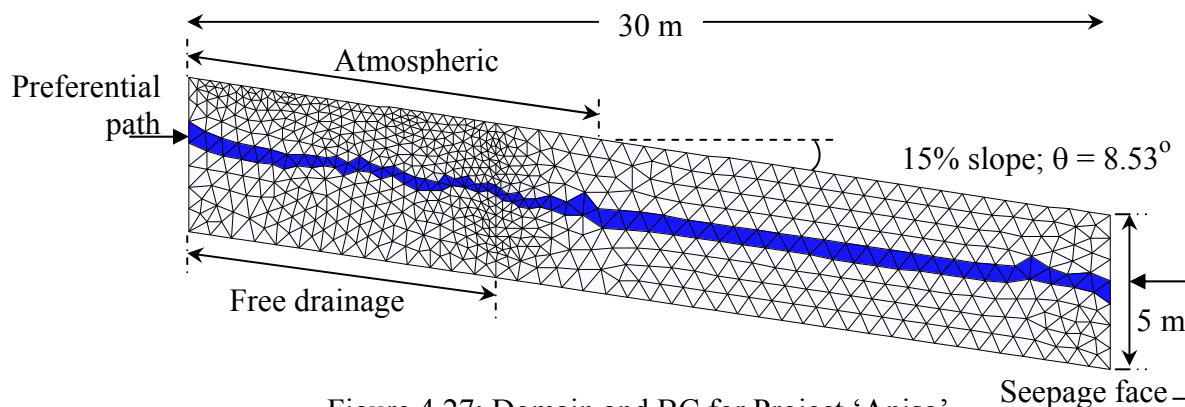
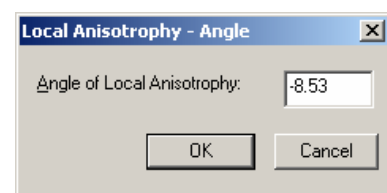
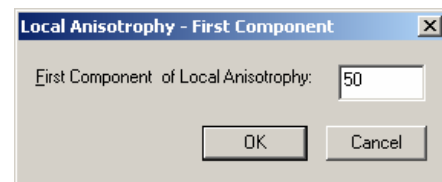


Figure 4.27: Domain and BC for Project ‘Aniso’

We define a local coordinate system for the anisotropy tensor oriented with the hill slope (Boundary Condition Editor/Local Anisotropy/Angle; **or** /Side bar-Angle: Select the entire domain and enter the angle as shown here).



We define the first component of the hydraulic conductivity tensor K_1 : Boundary Condition Editor/Local Anisotropy/First Component; **or** /Side bar- First Component: Select the elements representing the fault (approximately parallel to the slope; see Figure 4.27; remember that ‘Edit/Select by Rhomboid’ is a useful option here); enter the value for the first component K_1 as shown here. The second component is left at the default value (=1).



The orientation of the local coordinate system for the anisotropy tensor is shown in Figure 4.28. Note that z-axis is positive upwards.

We copy Project 'Aniso', and rename it 'Iso'; remove the anisotropy (set $K_1=K_2=1$) and run it in order to compare results from the two runs.

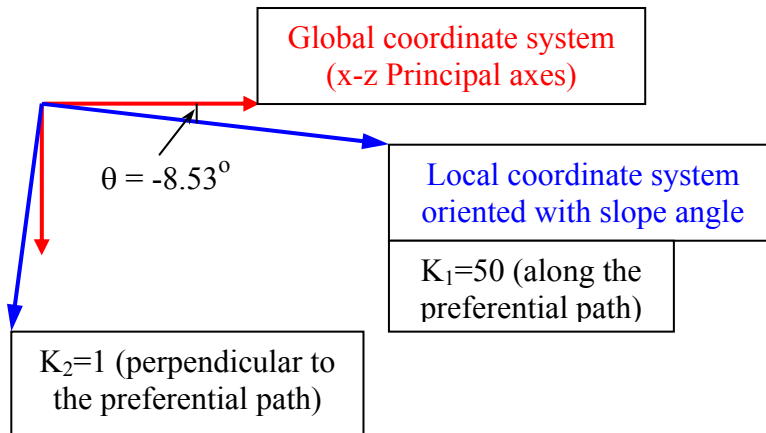


Figure 4.28: Coordinate system for local anisotropy

Results:

Figure 4.29 shows the velocity vectors at $T=1$ day. Note the direction and the magnitude of the velocity vectors along the preferential flow path. Figure 4.30 shows the velocity vectors for the isotropic domain (K_s is equal everywhere in the domain).

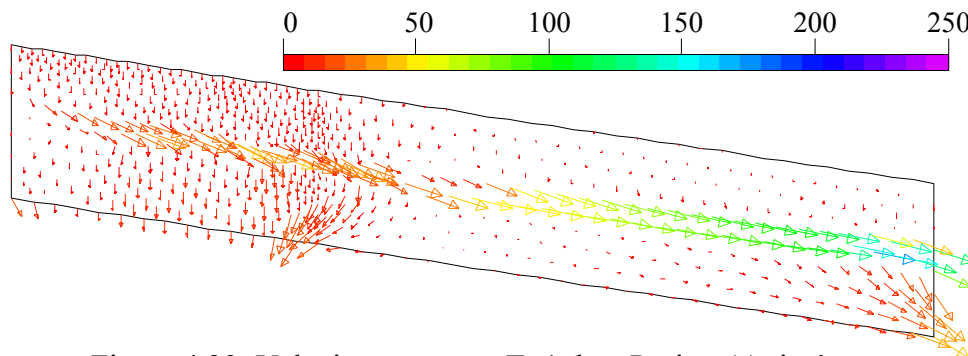


Figure 4.29: Velocity vectors at $T=1$ day; Project 'Aniso'

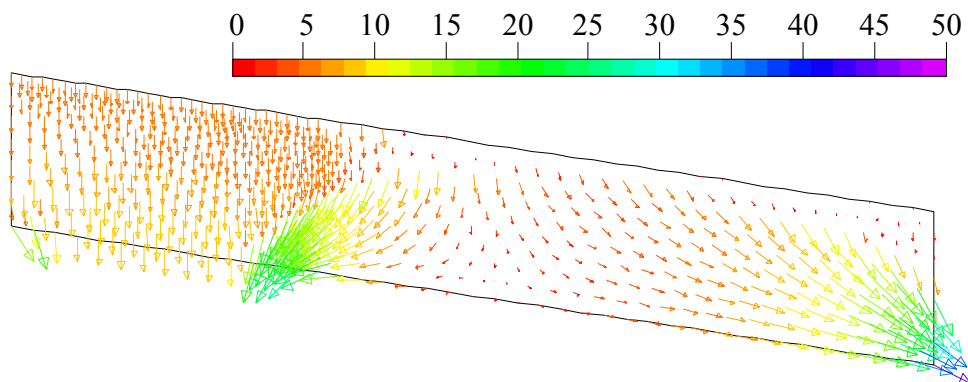


Figure 4.30: Velocity vectors at $T=1$ day; Project 'Iso'

Figure 4.31 shows the effect of the preferential path in lowering the pressure heads as a result of enhancing drainage.

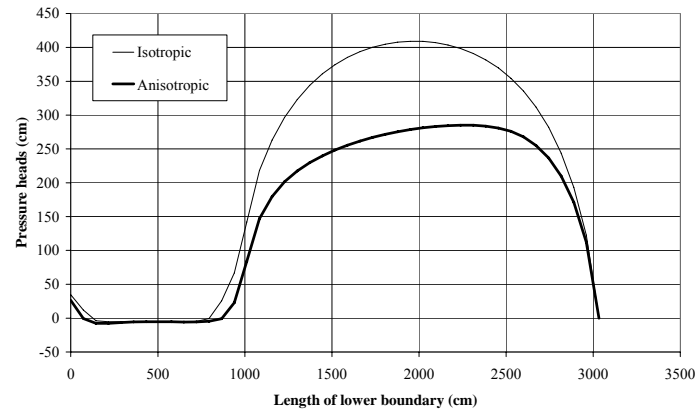


Figure 4.31: Pressure heads along the lower boundary for Projects 'Aniso' and 'Iso'

Figures 4.32 and 4.33 show how the preferential path slightly reduced the cumulative free drainage flux, but significantly increased the cumulative seepage face flux, respectively.

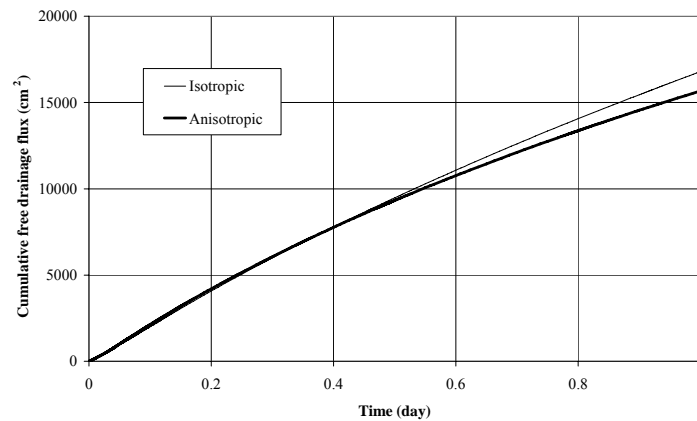


Figure 4.32: Cumulative free drainage flux for the Projects 'Aniso' and 'Iso'

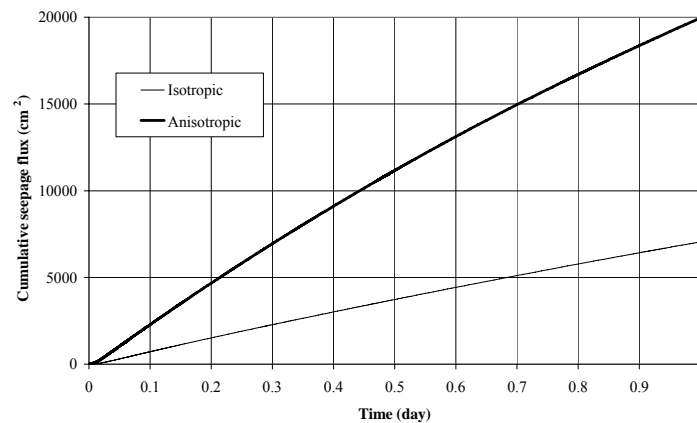


Figure 4.33: Cumulative seepage face flux for the Projects 'Aniso' and 'Iso'

4.9 Tension Disc Infiltrometer

The axisymmetric option in Geometry may be used to simulate a three-dimensional domain with symmetry around the vertical axis (z in HYDRUS). This option in HYDRUS creates a domain that is essentially still two-dimensional; it represents a volume that results from revolving the 2-D domain around the symmetry z -axis (refer to Section 1.1.2, Figure 1.2b). Practical applications include modelling flow for the following conditions:

- A single drip source.
- Finite-length cylindrical laboratory soil columns.
- Disc or ring infiltrometers.
- Wells

In this section we will demonstrate flow under a disc infiltrometer. In addition, we show the effect of introducing a confining ring under the source. We use the capillary barrier phenomenon to model the isolating effect of the ring.

Domain and boundary conditions:

The domain measures 50×50-cm (radius and depth, respectively). This means that we are modelling a 100×50-cm soil cylinder (diameter and depth, respectively). The disc infiltrometer has a 20-cm diameter membrane; therefore, it is represented by a 10-cm radius on top of the domain. All related Projects are found on the CD (**..\HYDRUS Manual Projects\Sect-4**).

The lower boundary condition is free drainage. The disc infiltrometer provides water at a constant negative head. However, the experiment is conducted in two stages:

0 - 0.1 day pressure head = -20 cm
 0.1 - 0.5 day pressure head = -5 cm

The pressure heads are entered as variable head data in the window “Variable Boundary Condition”, column “GWL” as discussed in Section 1.1.10-3.

The 20-cm diameter isolating ring that forces flow from the disc infiltrometer to be vertical could of course be modelled using a no-flux condition. As an exercise, we model the ring using the concept of capillary barriers. The subsurface for this purpose is represented by two materials of contrasting texture, fine and coarse. Figure 4.34 shows that the technique is effectively inhibiting any lateral flow (see location of observation points in Figure 4.35). The effect of the ring is investigated for two soils, namely, a silt and a loamy sand. Details of the domain and its boundary conditions are shown in Figure 4.35.

Alternative approach (use of Local Anisotropy)

The ring can be alternatively (and more precisely) simulated using only one material but by significantly decreasing the coefficients of the anisotropy tensor in certain locations. Refer to Project ‘DF-Aniso’ on the CD (**..\HYDRUS Manual Projects\Sect-4**).

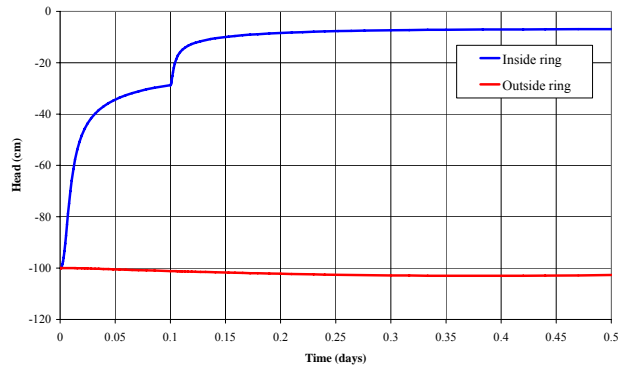


Figure 4.34: Effect of a capillary barrier in preventing lateral flow

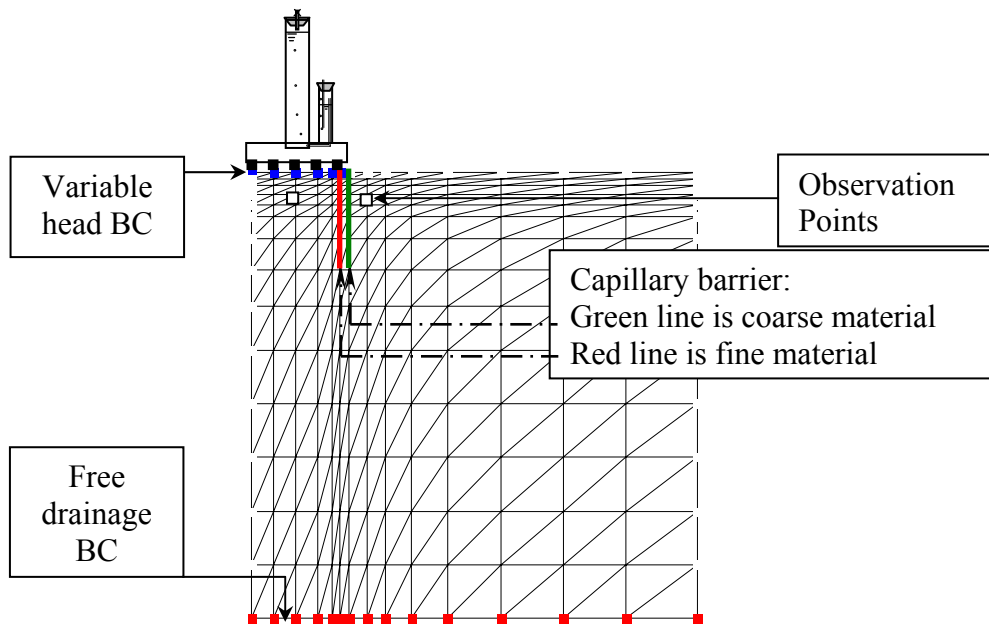


Figure 4.35: Domain and boundary conditions for axi-symmetric model

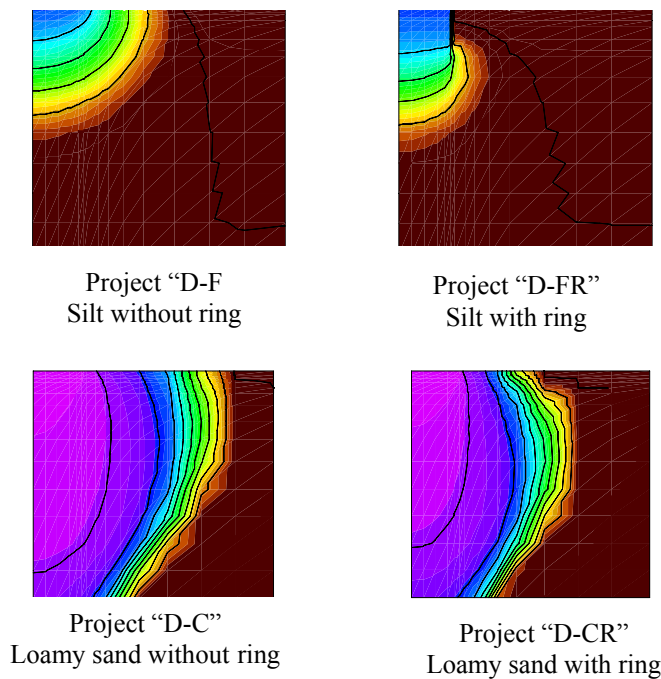


Figure 4.36: Advance of wetting front under disc infiltrometer

4.10 De-watering of a Cylindrical Pit

Project 'Dewat' simulates de-watering of a cylindrical pit having a diameter of 10 m and a depth of 8 m. A constant head BC is located at 25 m away from the edge of the pit. The initial water table is at a depth of 5 m; hydrostatic equilibrium is assumed. Water is pumped at a rate of 50 m³/day for a period of 4 days. The cross-section of the pit is shown in Figure 4.37.

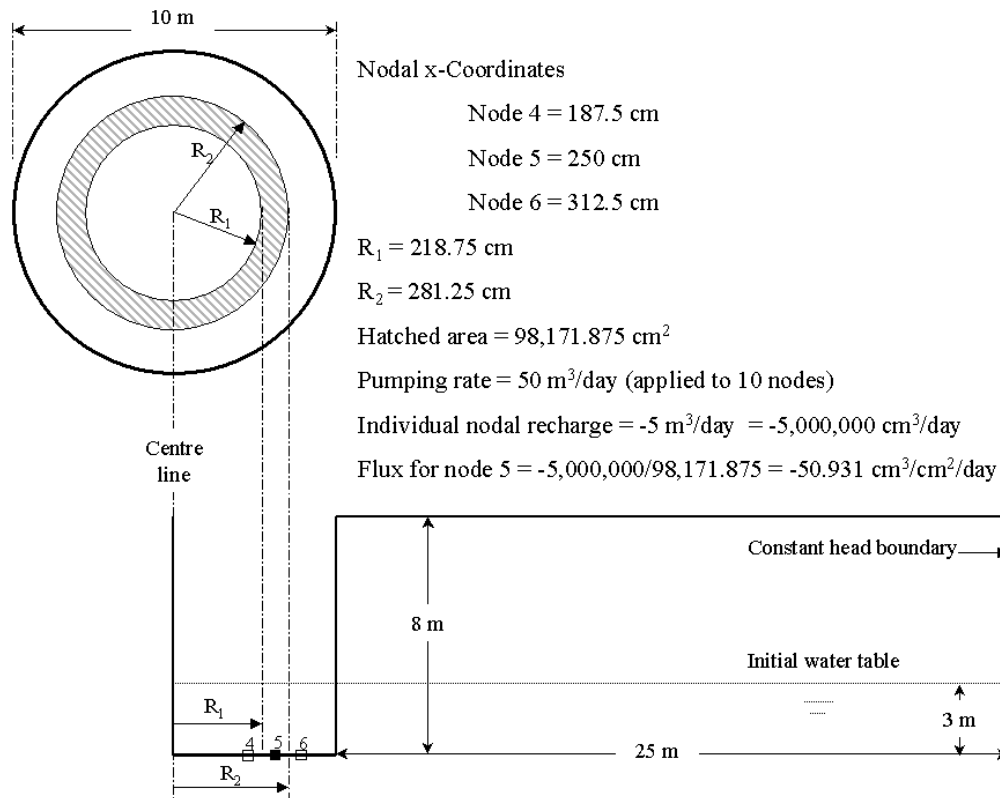
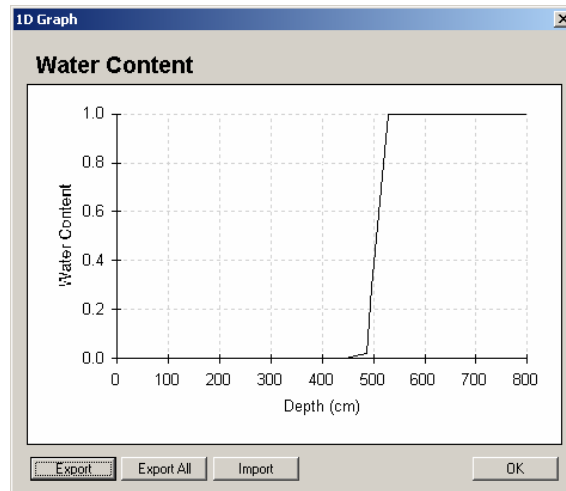


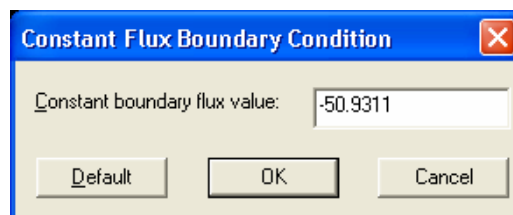
Figure 4.37: Sample calculation for nodal flux from recharge rate

The problem is modeled as axisymmetric vertical flow. The base of the pit is discretized into 10 nodes where each node is assigned a recharge rate of -5 m³/day (in Boundary Condition Editor/Condition/Nodal Recharge).

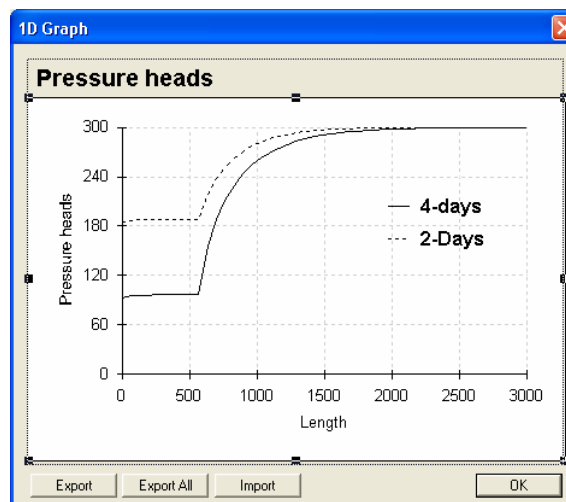
The space representing the excavated soil (the pit) is assigned a high-porosity (=0.999), high-conductivity material (=100 m/day) that mimics air (as in the case of the imaginary surface layer of example 4.7). The water content profile shown in the following page demonstrates that the technique works well; the figure shows a vertical cross-section through the pit prior to pumping (at Time=0). The water content between 500 and 800 cm (below the water table) in the excavated area is equal to unity (it represents water), but drops to zero just above the water table due to the high Alpha and n used. Project 'Dewat' is found on the CD (..\HYDRUS Manual Projects\Sect-4).



Note that once the recharge is assigned to the ten nodes at the base of the pit, they become constant-flux boundary nodes; HYDRUS assigns the appropriate value of the flux to each node. Select the 'Constant Flux' option in 'Boundary Condition Editor', left-click on node 5 twice, and the flux value will be displayed as shown below. Figure 4.37 demonstrates how this flux is calculated.



The pressure heads at the base of the domain are shown below; they demonstrate the water table depression as pumping progresses.



4.11 Steady-State Horizontal Flow

Project 'Hor-Flow' uses the horizontal-plane flow option. This option is mostly relevant only to saturated water flow. The horizontal flow domain shown in Figure 4.38 is a 'Plan view' where the third dimension represents soil depth. Figure 4.38 shows the constant head boundary conditions imposed at the two corners of the flow domain. Project 'Hor-Flow' is found on the CD (**..HYDRUS Manual Projects\Sect-4**).

Steady-state head contours are shown in Figure 4.38 (Graphical Display of Results/Options/Isolines Parameters/Increment=1).

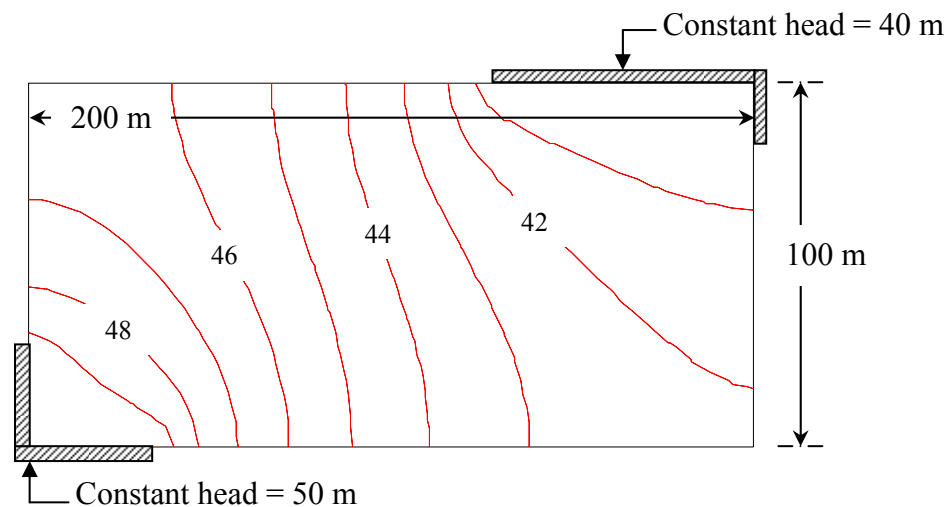


Figure 4.38: Flow domain, boundary conditions, and steady-state head contours

A 3D surface representing the steady-state water table can be obtained as follows. Obtain 11 1-D graphs at each 10 m of the y-coordinate from the graphical interface and export them into text format. Next, import the data into a software package such as 'Surfer' to plot a 3-dimensional surface (see Figure 4.39).

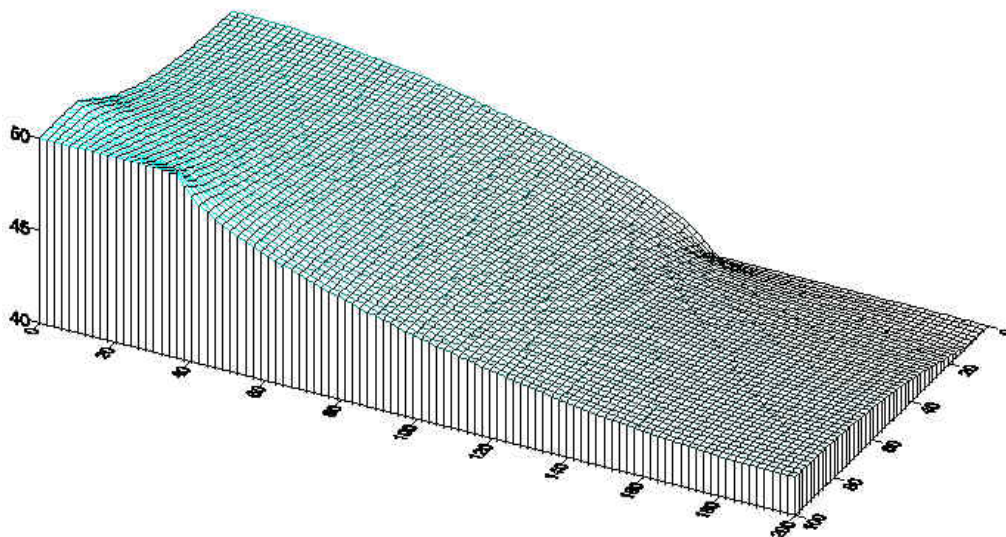


Figure 4.39: 3-D surface representing steady-state water table

4.12 Print Times > 250

The graphical interface (Print Information/Number of Print Times) allows one to enter up to 250 print times. However, the HYDRUS code (h2d_calc.exe) and the graphical display interface (Post-Processing/Graphical Display of Results) allow print times of up to 5000. This is only possible in HYDRUS version 2.007 and later.

The user has to enter the print times that exceed 250 manually by accessing the “Selector.in” input file using a spreadsheet. Below is the relevant part of the input file. Project “Trial” is found on the CD (..\HYDRUS Manual Projects\Sect-4).

Relevant part in Input file “Selector.in”

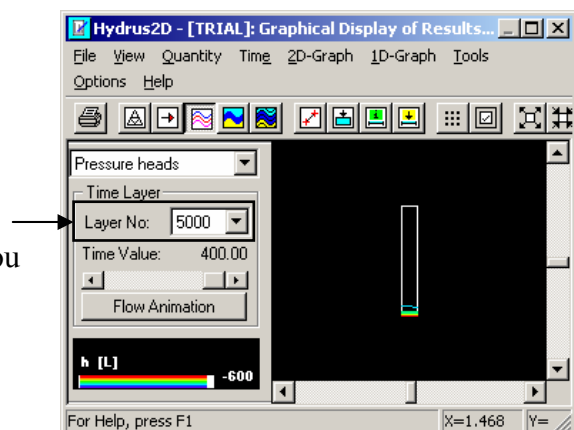
dt	dtMin	dtMax	DMul	DMul2	ItMin	ItMax	MPL
0.001	0.001	1	1.3	0.7	3	7	5000
tInit	tMax						
0	400						
0.08	0.16	0.24	0.32	0.4	0.48		
0.56	0.64	0.72	0.8	0.88	0.96		

- Under “**MPL**”, enter the number of print times (maximum 5000).
- Print times are entered in rows each containing 6 print times. Make sure that the number of entered print times add up to MPL.
- Save the file; if you are use in Excel
 - Click the “Save” button
 - The “Save as” window will appear
 - Keep the default file name “Selector.in”
 - Accept the option to overwrite the existing file
 - Click “Yes” to keep the file in the existing format.

Open the project in HYDRUS; run HYDRUS without accessing the “Print Information” window.


Project “Trial”

Click on “Graphical Display of Results” and you will see 5,000 time layers as shown here.



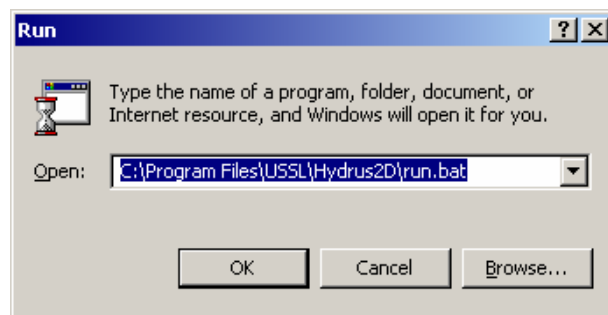
4.13 Running Multiple Sequential Simulations

If you have a large number of simulations that need to be run sequentially, you can do this from the Start-Run option in Windows, which is similar to the old DOS prompt. The multiple runs are executed from a batch file. In this example, we want to run Projects “Trial1” and “Trial2”. Follow the steps below:

1. Identify the directory path of the project you want to run; in this case it is: “D:\HYDRUS manual\trial1”. Create a text file (e.g. in Notepad) and give the file a name; in this case “Path1.txt”; copy the path into the file and save.
2. Repeat this process. Copy the path “D:\HYDRUS manual\trial2” and save the file as “Path2.txt”.
3. When Calculate/Run HYDRUS () is clicked it executes the h2d_calc.exe program. When the run finishes it requires one to press “Enter” on the keyboard. Accordingly, we must have a text file that contains the hexadecimal code for “Enter”. Create a text file in Notepad; write in it “1C” (code for Enter), call the file “Return.txt”, and save.
4. Create the batch file. First we should let HYDRUS know where the location of the project is; this is done by copying the text file that contains the path and renaming it “level_01.dir” (the file that HYDRUS looks for when not run from the graphical interface to identify the directory where the project exists). Create the following file to run projects “Trial1” and “Trial2”. Call the file “Run.bat”; notice that the extension of the file must be (.bat). Also notice that spaces are present between the DOS commands.

```
copy path1.txt level_01.dir
h2d_calc <return.txt
copy path2.txt level_01.dir
h2d_calc <return.txt
```

5. Place the four files in the HYDRUS home directory (where the “h2d_calc.exe” is stored; usually C:\Program Files\USSL\Hydrus2D); the files in this case are: “Run.bat”, “Path1.txt”, “Path2.txt”, and “Return.txt” (copies of these files are found on the CD; (..\HYDRUS Manual Projects\Sect-4\4_13)).
6. In Windows: Start/Run Through “Browse”, go to the HYDRUS home directory and click on the “Run.bat” file; then click OK.



5. INVERSE SOLUTION

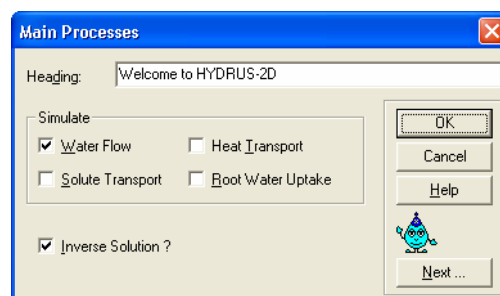
5.1 Inverse Solution Options in HYDRUS

Main Processes

In normal simulations or “forward problems”, the modeller provides the known input parameters and HYDRUS runs once through the specified time duration.

However, under the “Inverse Solution” option, HYDRUS runs the problem using the initial estimate for the parameters, compares the simulation results with observed experimental data that the user

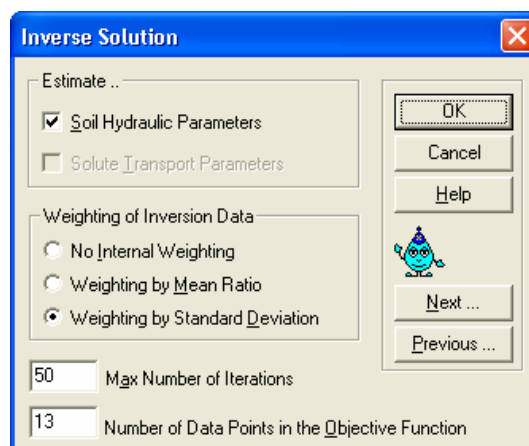
provides, and then re-runs the model a number of times with modified sets of parameters until a close fit to the observed experimental data is obtained. The number of iterations depends, among other things, on how close the initial input parameters were to the final estimate. This process is usually referred to as “Model calibration”, or “Inverse parameter estimation”. The technique has many valuable applications, which will be discussed in a later section. For more details, refer to Appendix V.



Click next.....

Inverse Solution – This window appears only if you have checked “Inverse solution?” in “Main Processes.

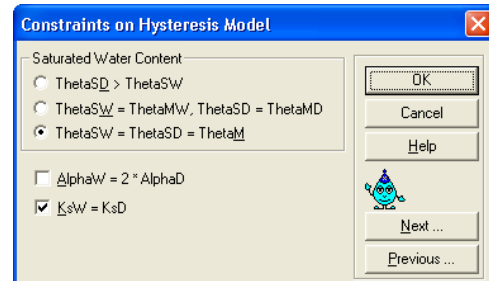
- **Estimate:** Select whether you want to estimate soil hydraulic parameters, solute transport parameters, or both. The latter will only be activated if solute transport is simulated (mostly outside the scope of this document).
- **Weighting of inversion data** provides the user with two different options for weighting the inversion data. The weighting process manages differences in weights because of the use of different types of calibration data. Refer to Appendix V for more details.
- **The maximum number of iterations** is an upper limit for the number of runs if convergence is not achieved. In such cases the user might change the starting values for the optimised parameters or limit their number. If zero is entered, the simulation will run once (as in usual forward modelling). The latter option is very useful since it gives you an option to test the quality of the initial input parameters. HYDRUS will then display the observed and predicted values under “Observation Points” and by means of other graphical outputs.
- **The number of data points in the objective function** refers to the number of observed laboratory and/or field data in the optimisation such as pressure head water content, and/or flux. For example, let us assume you conducted a 10-hour drainage



experiment in a laboratory soil column and recorded 10 water content readings using a TDR device, one reading every hour. You want to use the inverse solution option to estimate the soil hydraulic parameters that would give the closest prediction to those laboratory measurements. You should enter 10 as the number of data points, and you'll be prompted to enter the time and the 10 water content values under "Data for Inverse Solution".

If hysteresis is checked under 'Soil Hydraulic Model', then the following window will appear. You can apply constraints on the hysteresis model.

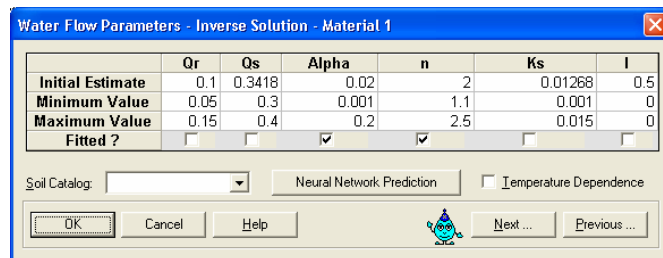
- Θ_{SD} and $\Theta_{SW} = \theta_s$ for drying and wetting cycles, respectively.
- $\Theta_M = \theta_m$ the fictitious water content in the modified van Genuchten model (MD and MW are for main drying and main wetting, respectively).
- α_D and α_W represent α for drying and wetting cycles, respectively. More details are found in Šimůnek et al. (1999).



This 'Water Flow Parameters' window will now appear.

Notice that there are check boxes beneath each parameter. Checking a box along the line 'Fitted?' means that the particular parameter will be optimised. You can enter

minimum and maximum values to make the search process more effective. If you do not about those limits, leave zeros in the particular boxes.



The meaning of the various inputs required in this window is well explained in the on-line Help. We will explain all available options here. Refer to Table 5.1 for an explanation of the window shown here.

	X	Y	Type	Position	Weight
1	1	-2	0	1	1
2	2	3	0	2	1
3	3	-2	0	3	1
4	4	-5	0	4	1
5	5	0.5	0	5	1
6	6	7	0	6	1
7	10	5.5	1	1	1
8	20	0.265	2	1	1
9	30	0.01	3	1	1
10	400	0.16	2	0	1
11	-5	0.278	5	1	1
12	-20	0.0015	6	2	1
13	10000	0.0023	7	1	1

Remember that data points containing time (i.e., type=0 through 4) must be entered first before any other data (such as retention or conductivity data or prior information).

Table 5.1: Details corresponding to window 'Data on Inverse Solution' (page 5.2)

	x	y	type	Position	wt
1	Time= 1 day	Observed inflow value= -2	(0) code for Cumulative flux	(1) code for constant pressure boundary	1
2	Time= 2 days	Observed outflow value= 3	(0) code for Cumulative flux	(2) code for seepage face boundary	1
3	Time= 3 days	Observed inflow value=-2	(0) code for Cumulative flux	(3) code for variable pressure boundary	1
4	Time= 4 days	Observed inflow value=-5	(0) code for Cumulative flux	(4) code for atmospheric boundary	1
5	Time= 5 days	Observed outflow value=0.5	(0) code for Cumulative flux	(5) code for drain boundary	1
6	Time= 6 days	Observed outflow value=7	(0) code for Cumulative flux	(6) code for free drainage boundary	1
7	Time= 10 days	Observed pressure head value=5.5 cm	(1) code for pressure head	(1) Observation point number	1
8	Time= 20 days	Observed water content value= 0.265	(2) code for water content	(1) Observation point number	1
9	Time= 30 days	Observed flow value= 0.01	(3) code for boundary flow	(1) code for constant pressure boundary	1
10	Time (400days)	Observed average water content value= 0.16	(2) code for water content	(0) code indicating that observed water content is for whole domain	1
11	Observed pressure head value=-5 cm	Observed water content value= 0.278	(5) code for point on water retention curve	(1) code representing material number	1
12	Observed pressure head value=-20 cm	Observed hydraulic conductivity value= 0.0015	(6) code for point on hydraulic conductivity function	(2) code representing material number	1
13	dummy (10000)	Observed Alpha parameter value= 0.0023	(7) code for parameter Alpha	(1) code representing material number	1

5.2 Inverse Modelling of Controlled Laboratory Column

In this section we will explore inverse parameter estimation by modelling flow through a controlled laboratory column. We highlight cases that demonstrate the following aspects:

- Non-uniqueness of the inverse solution
- Temporal and spatial distribution of calibration data
- Effect of initial parameters
- Correlation of variables and statistical issues
- Validation of results

We also provide some guidelines that you should keep in mind when using this technique.

In this series of simulations we use data from Rassam and Williams (1999a). First we give a brief description of the experiment: a 13-cm tailings sample is packed into a flow cell where transient water contents and initial and final pressure heads are measured. The lower boundary is a constant suction (negative head) and the upper boundary is a constant flux. Two wetting cycles covering different suction ranges are simulated in the same flow cell containing the same soil sample (wetting cycles 'a' and 'b'; see Figure 7 of paper).

The water retention parameters obtained from the inverse solution and the transient data are compared against those measured experimentally under static conditions in the same flow cell. In addition, unsaturated hydraulic conductivities predicted from the water retention curve are also compared to the experimental measurements. All related projects are found on the CD (**HYDRUS Manual Projects\Sect-5**).

5.2.1 Simulating Wetting Cycle 'a'

Refer to Project series 'a1-a8' on the CD. The initial and boundary conditions are:

- Initial pressure head = -780 cm
- Lower BC is a constant head = -780 cm
- Upper BC is a constant flux = 0.00643 cm/min (=3.86 mm/hr)
- $Q_s=0.3418$; $Q_r=0.1$

Project 'a1'

Calibration data = 12 water content measurements.

Optimise Alpha, n, and Ks.

Project 'a2'

Calibration data = 7 water content measurements covering the same range as in Run 'a1' (i.e. every other point was eliminated from the data set).

Optimise Alpha, n, and Ks.

Comments on Projects 'a1' and 'a2':

Figures 5.1, 5.2, and 5.3 show that runs 'a1' and 'a2' produced almost identical results. Apparently the number of calibration points did not matter as long as the temporal distribution was similar. The optimised parameters were found to be highly correlated.

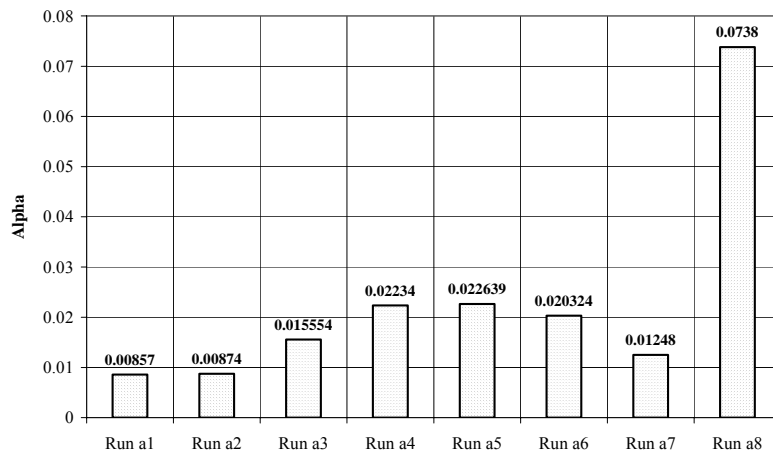


Figure 5.1: Comparison of parameter Alpha for Projects 'a1-a8'

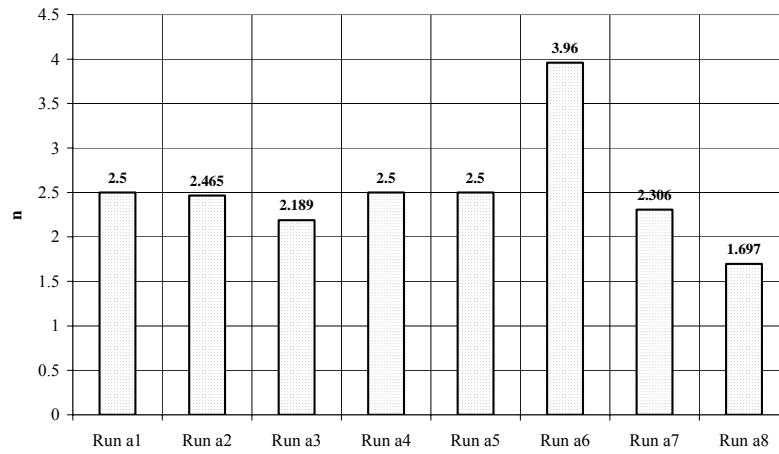


Figure 5.2: Comparison of parameter n for Projects 'a1-a8'

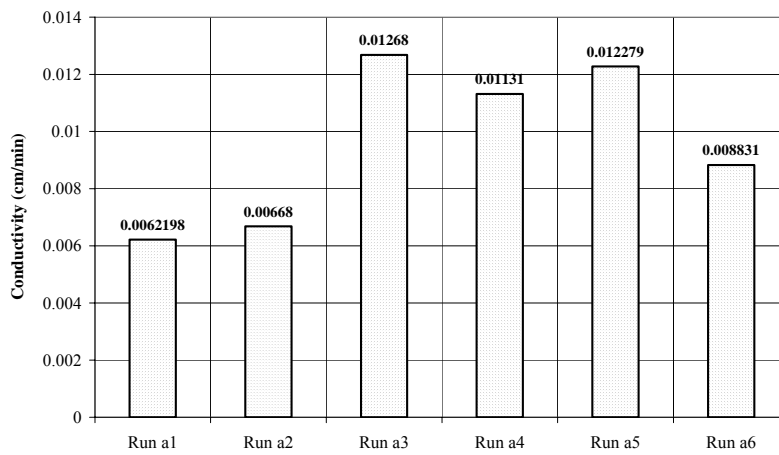


Figure 5.3: Comparison of parameter Ks for Projects 'a1-a6'

Project 'a3'

Calibration data = 12 water content measurements and a final pressure head at the surface.

Optimise = Alpha, n, and Ks.

Comments on Projects 'a1' and 'a3':

Figures 5.1, 5.2, and 5.3 show that although Projects 'a1' and 'a3' use identical water content calibration data, the additional calibration point (pressure head) had a significant impact on the results.

Figure 5.4 shows that incorporating the pressure head at the end of the simulation greatly impacted the simulated heads at time > 100 minutes and produced better agreement with the measurement.

Figure 5.5 shows that the simulated water contents for Projects 'a1' and 'a3' are very similar; both match the observed values. However, a close look at the 'Inverse Solution' results in 'Post-processing' (or in the Fit.out output file) shows that the R^2 is lower for Project 'a1'.

Adding the additional calibration point in Project 'a3' (final pressure head) resulted in much lower correlation coefficients between the parameters compared to Project 'a1'.

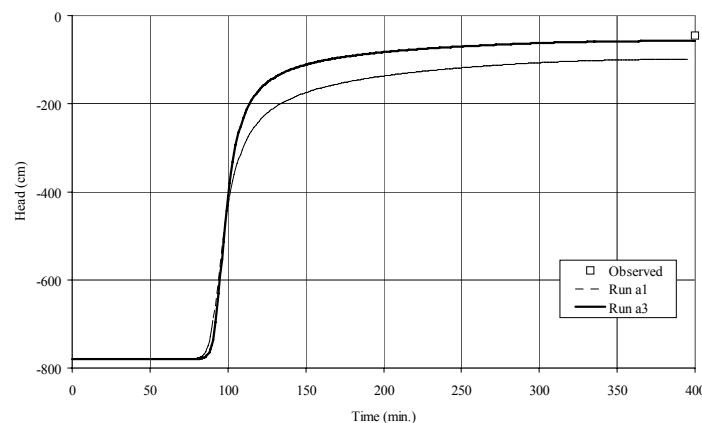


Figure 5.4: Comparison of simulated head for Projects 'a1' and 'a3'

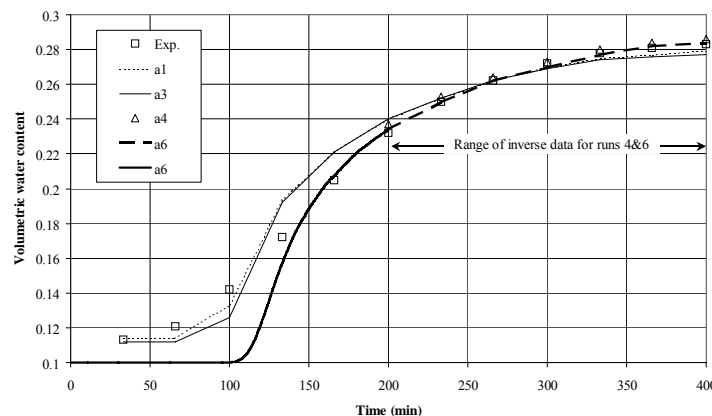


Figure 5.5: Comparison of simulated and observed water contents for Projects 'a-1', 'a-3', 'a-4', and 'a-6'

Project 'a4'

Calibration data = 7 water content measurements but now at times > 200 minutes (i.e., the calibration is biased toward the wet end of the curve).

Optimise = Alpha, n, and Ks.

Comments on Project 'a4':

Figures 5.1, 5.2, and 5.3 show that the process resulted in a different set of parameters.

Figure 5.5 shows good agreement between observed and simulated water contents (in the calibrated wet range, time > 200 min).

Project 'a5'

Calibration data and parameter optimisation = As in Project 'a4'.

Apply a more stringent water content tolerance.

Comments on Project 'a5':

Figures 5.1, 5.2, and 5.3 show that using a more stringent tolerance slightly impacted the hydraulic conductivity.

The run time was doubled.

Project 'a6'

Calibration data and parameter optimisation = As in Project 'a4'.

Different set of initial parameter values Alpha, n, and k.

Comments on Project 'a6':

Figures 5.1, 5.2, and 5.3 show that the optimised parameters are now significantly different.

Figure 5.5 shows that good agreement was obtained between simulated and observed water contents in the calibrated wet range.

However, the water content predictions diverged from the observed values at the beginning of the simulation ($0 < \text{time} < 200$) for which no calibration data was used.

Project 'a7'

Calibration data = As in Project 'a1'.

Optimise Alpha and n (Ks excluded).

Assume that a good initial estimate for the hydraulic conductivity was provided.

Comments on Project 'a7':

Figures 5.1, 5.2, and 5.3 show that the optimised parameters are different from those obtained in Project 'a1'.

The correlation coefficient was reduced to -0.2954.

Project ‘a8’

Calibration data and parameter optimisation = As in Project ‘a7’.
A poor initial estimate for the hydraulic conductivity was provided.

Comments on Project ‘a8’:

Figures 5.1, 5.2, and 5.3 show that the optimised parameters are now significantly different from those obtained in Project ‘a1’.

The correlation coefficient was low at 0.2247.

Validation of Inverse Solution Results for Projects ‘a-‘

The eight previous simulations produced eight sets of results. Which one is the best? Fortunately, we have two independent data sets against which we can validate our results. Firstly, a wetting water retention curve, and, secondly, a number of unsaturated hydraulic conductivities measured over a limited suction range. Both data sets were experimentally obtained from the same experiment, which means that they can be given a high degree of confidence.

The optimised parameters obtained from the inverse solution are used to plot the corresponding water retention curves. They are compared with the experimental curve. Results from simulations ‘a2’ and ‘a5’ are not shown because they are similar to simulations ‘a1’ and ‘a4’, respectively. Refer to Figure 5.6:

Simulation ‘a1’ (12 water content data points) resulted in an over-prediction of the water contents (curve shifted upwards).

Simulation ‘a3’ (12 water content data points and final head) provided the best fit to the experimental measurements.

Simulation ‘a4’ (7 water content data points at time>200) resulted in an under-prediction of the water contents (curve shifted downwards).

Simulation ‘a6’ (as in simulation ‘a4’ but different initial parameters) also resulted in a poor fit to the experimental values.

Simulation ‘a7’ (as in ‘a1’ except that Ks was not optimised; a good estimate was available) resulted in a better prediction than that provided by simulation ‘a1’. Therefore, provided the estimate for Ks was good, excluding Ks from the optimisation procedure gave a better solution with relatively low correlation coefficients.

Simulation ‘a8’ (as in simulation ‘a7’ but with a poor initial estimate for Ks) produced poor results.

Figure 5.7 shows similar trends when unsaturated hydraulic conductivities predicted with the water retention parameters obtained from Projects ‘a1-a8’ are compared to the experimental data. Project ‘a3’ still gives the best fit to the measured hydraulic conductivities.

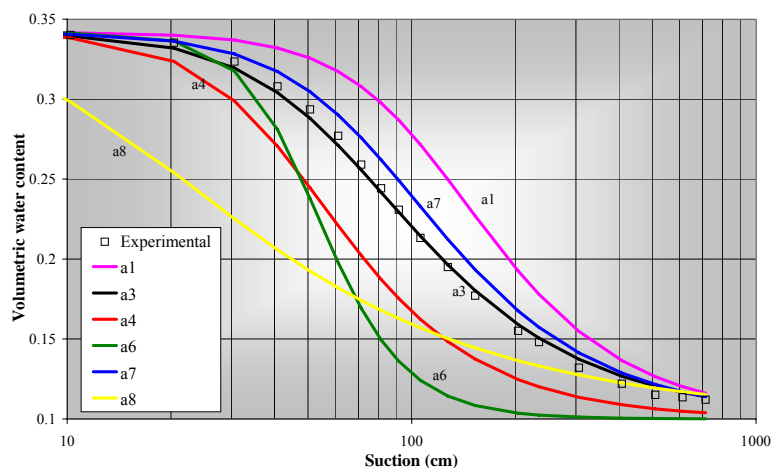


Figure 5.6: Validation of water retention parameters obtained from Projects ‘a1-a8’

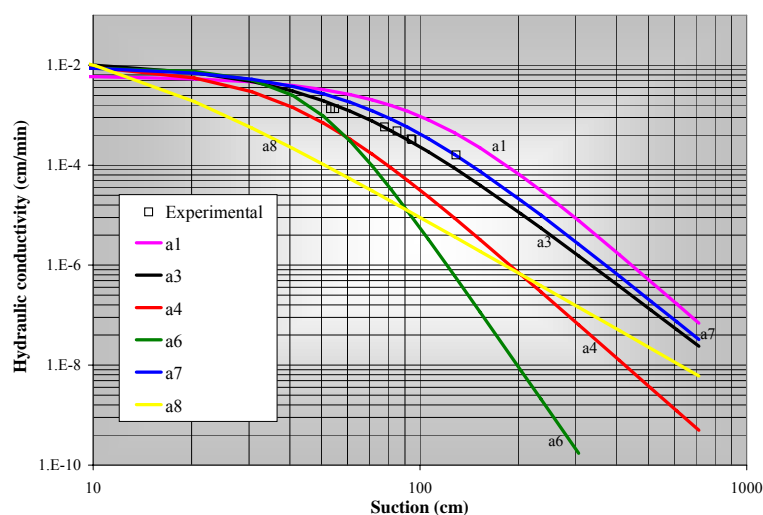


Figure 5.7: Validation of unsaturated hydraulic conductivity data predicted using water retention parameters obtained from Projects ‘a1-a8’

Concluding Comments on Project Series ‘a-‘:

Note that all simulations produced a good visual fit to the experimental transient water content data. Therefore, a good visual fit does not ensure that the correct results are obtained; one should closely look at the statistical information provided in the ‘Fit.out’ file.

Having additional data can be very advantageous (e.g., the final pressure head in simulation ‘a3’). This may significantly reduce the correlation coefficients.

If you have a good experimental measurement of a certain parameter (e.g., K_s in our example), it is advantageous not include that parameter in the optimisation. This may reduce correlation between the remaining parameters and lead to a better solution. However, if the initial estimate is poor, this may have the opposite effect and adversely affect the solution. Hence, it is crucial to know “how much you trust the parameters”.

Initial parameters have an impact on the simulation results if the solution is non-unique. This phenomenon will be further investigated in Section 5.4

5.2.2 Simulating Wetting Cycle ‘b’

Refer to Project series ‘b1-b6’ on the CD. The initial and boundary conditions for this series are:

- Initial head = -208 cm
- Lower BC is a constant head = -208 cm
- Upper BC is a constant flux = 0.008467 cm/min (=5.08 mm/hr)
- $Q_s=0.3418$; $Q_r=0.1$

Project ‘b1’

Calibration data = 6 water content measurements.

Optimise Alpha, n, and K_s .

Comments on Project ‘b1’:

Figure 5.8 shows that the calibration produced an excellent fit to the observed values. The optimised parameters are plotted in Figures 5.9, 5.10, and 5.11.

Correlation coefficients are high.

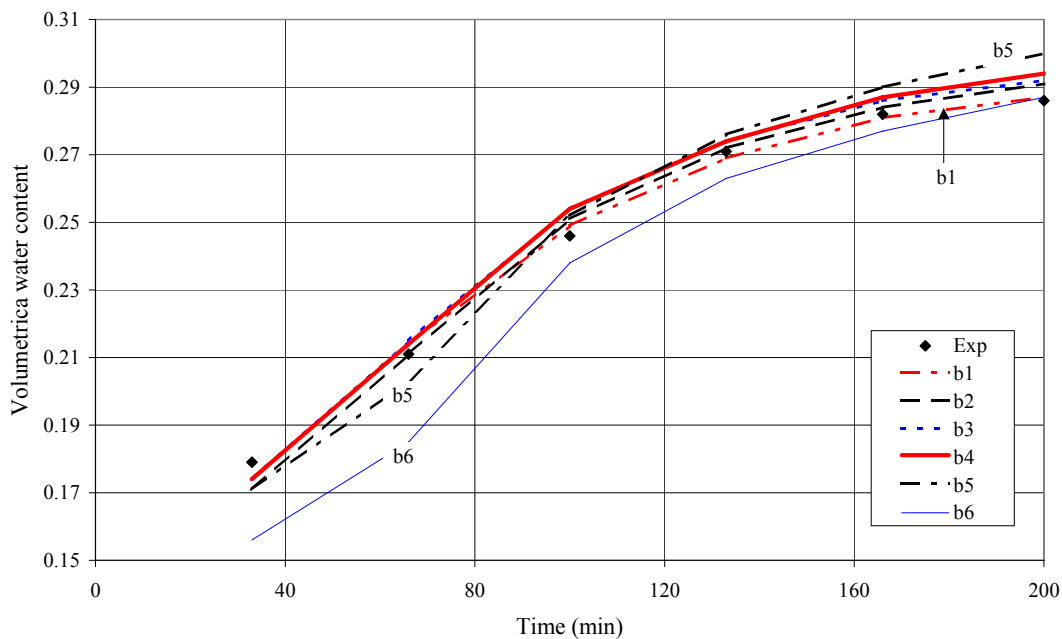


Figure 5.8: Comparison of simulated and observed water contents for Projects ‘b-1’ through ‘b-6’.

Project ‘b2’

Calibration data = 6 water content measurements and final pressure head.

Optimise Alpha, n, and K_s .

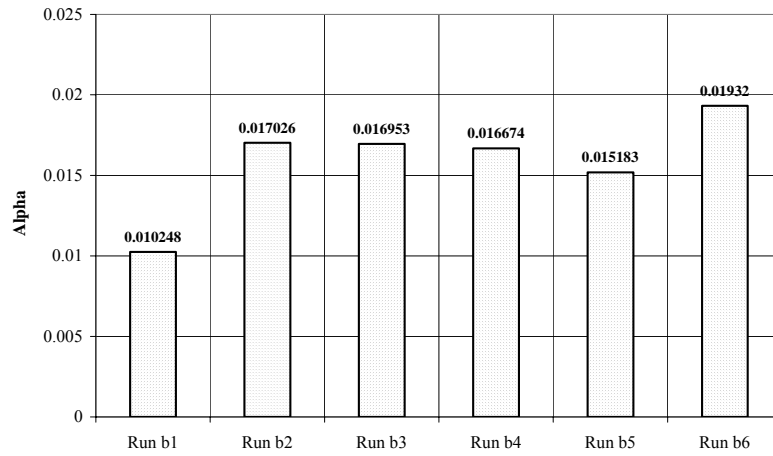


Figure 5.9: Comparison of parameter Alpha for Projects 'b1-b6'

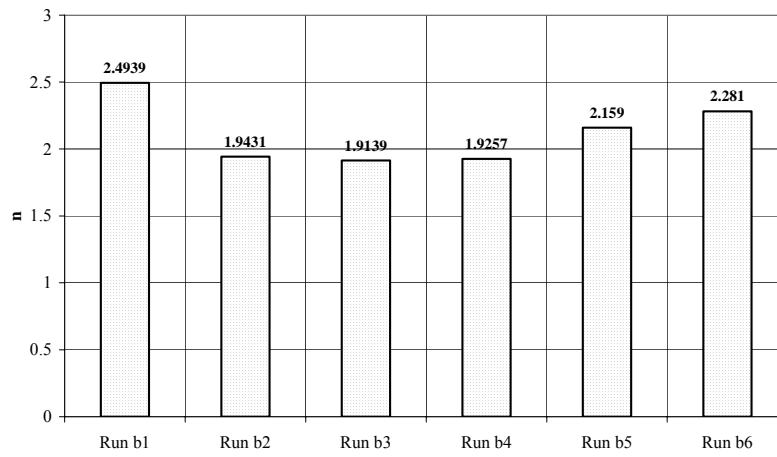


Figure 5.10: Comparison of parameter n for Projects 'b1-b6'

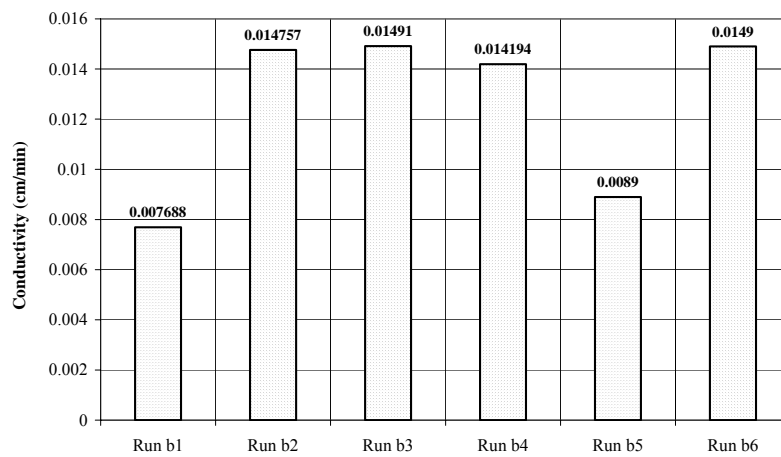


Figure 5.11: Comparison of parameter Ks for Projects 'b1-b6'

Project 'b3'

Calibration data = 6 water content measurements and 2 unsaturated hydraulic conductivity measurements (weight=2 for the K measurements).

Optimise Alpha, n, and Ks.

Project 'b4'

Calibration data = 6 water content measurements, final pressure head, and 2 unsaturated hydraulic conductivity measurements (weight=2 for K and the pressure head measurements).

Optimise Alpha, n, and Ks.

Comments on Projects 'b2', 'b3', and 'b4':

Figure 5.8 shows that the calibration resulted in an excellent fit to the observed values for the three Projects 'b2-b4'. The optimised parameters are plotted in Figures 5.9, 5.10, and 5.11.

The optimised parameters for the three projects are in close agreement. Adding additional calibration data types (pressure heads or hydraulic conductivities) was not particularly advantageous in this example. Results would have improved if we optimised more parameters.

Correlation coefficients are slightly lower for Project 'b2'.

Correlation coefficients remained high for Project 'b3'.

Correlation coefficients are much lower for Project 'b4'.

Project 'b5'

Calibration data = 6 water content measurements, final pressure head, 2 unsaturated hydraulic conductivity data, and one point on the dry end of the water retention curve (weight=2 for K, the pressure heads, and point on water retention curve).

Optimise Alpha, n, Ks, **and Qr**.

Project 'b6'

Calibration data = 6 water content measurements, final pressure head, 2 unsaturated hydraulic conductivity data, and a point on the dry end of the water retention curve (weight=2 for K and the pressure heads; weight=5 for point on water retention curve).

Optimise Alpha, n, Ks, **and Qr**.

Comments on Project ‘b5’ and ‘b6’:

Figure 5.8 shows that optimising Q_r caused skewing of the calibration for Project ‘b5’. For calibration point no. 9 (type 5; head=-306 cm, water content=0.132), the fitted water content was 0.151 (see file ‘Fit.out’ for Project ‘b5’).

For Project ‘b6’, putting a higher weight on the water retention data point (calibration point no. 9) resulted in a better estimate for the water content (predicted=0.142; observed=0.132). However, Figure 5.8 shows that this resulted in a worse fit to the transient water content data.

Validation of Inverse Solution Results for Projects ‘b-‘

Simulation ‘b1’: Figure 5.12 shows that calibration against water content data alone produced the worst fit to the observed values.

Simulations ‘b2-b4’: Figure 5.12 shows that adding an additional data type (final pressure head or K_s values) improved the calibration results. However, including both did not improve the results much further, that is, the extra information was redundant but had the advantage of reducing the correlation coefficients.

Simulations ‘b2-b4’: The three simulations failed to provide a good description of the dry end of the curve (suction > 100 cm). This is attributable to the fact that the pressure heads covered in the physical model (wetting cycle b) cover the wetter end of the curve (starting head=-208 cm). Note that we had a better overall prediction of the water retention curve in first set of simulations (wetting cycle a; simulation ‘a3’) because the pressure heads covered the dry end of the curve (the starting pressure head was -780 cm).

Simulation ‘b5’: Figure 5.12 shows that including a point on the dry end of the water retention curve (head=-306; water content=0.132) resulted in a better agreement towards the dry end of the water retention curve.

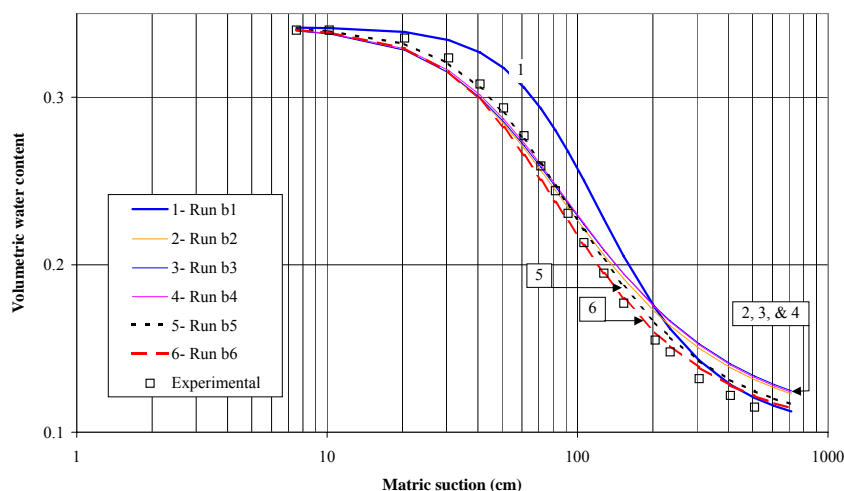


Figure 5.12: Validation of water retention parameters obtained from Projects ‘b1-b6’

Simulation ‘b6’: Figure 5.12 shows that increasing the weight (to 5) for the point on the dry end of the water retention curve (pressure head=-306;water content=0.132) improved the agreement towards the dry end of the water retention curve. However, the prediction was now worse in the wet range of the curve.

Concluding Comments on Project Series ‘b-‘:

Adding an extra data type improved the model predictions, but further additions seemed redundant.

Due to the fact that the experimental wetting cycle covered the wet end of the water retention curve, inferior prediction of the dry end of the curve resulted; wetting cycle ‘b’ provided a worse prediction than wetting cycle ‘a’.

Including a point from the water retention curve in the calibration process improved the fit in the dry end of the curve. However, putting a high weight on this data point resulted in a much poorer prediction of the wet end of the curve.

5.2.3 Optimising Parameter L in the Hydraulic Conductivity Function

This is the tortuosity factor in the hydraulic conductivity function. Its value was found to be 0.5 by Mualem, but there is no general agreement about this in the literature.

Parameter L is an input parameter in HYDRUS and can hence be optimized.

The shape of the hydraulic conductivity function is controlled mainly by the retention curve parameters ‘alpha’ and ‘n’. The saturated hydraulic conductivity merely scales the hydraulic conductivity function; it provides us with absolute values for the conductivity rather than only relative values provided by the analytical function.

Optimizing the tortuosity parameter permits one to change the shape of the hydraulic conductivity function independently of the retention function. Since the older version of HYDRUS could not jump over zero, we conduct below two runs, one starting with a positive L value and one with a negative L value.

Projects “L-Par” and “L-Par-ve”:

In these Projects we use the problem modeled in series “a-“. Let us use the optimized Alpha and n from Project “a3”, which resulted in the best fit to the water retention data. We calibrate against six measured unsaturated hydraulic conductivity values, in addition to the measured final pressure head (water contents are excluded). The hydraulic conductivity and the tortuosity factors are optimized.

Table 5.2: Optimization results of Projects “L-Par” and “L-par-ve”

	Parameter L	K_{sat} (cm/minute)
Project “a3”	0.5 (not optimised)	0.01268
Project “L-Par”	2.4×10^{-6} (optimised)	0.00969
Project “L-Par-ve”	-0.137 (optimised)	0.00894

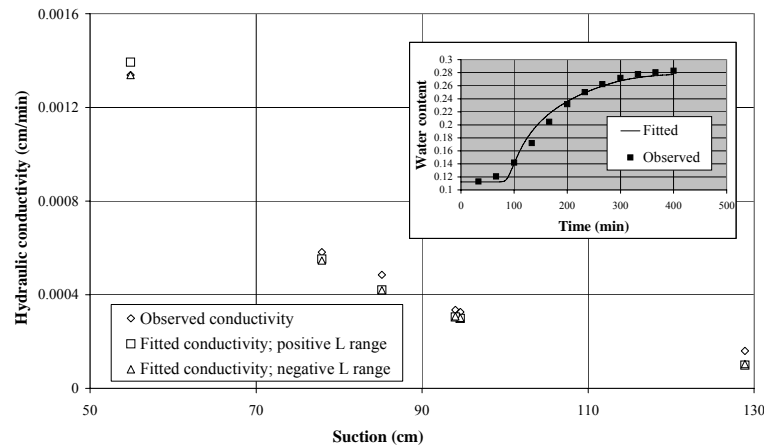


Figure 5.13: Measured and fitted conductivity for Projects “L-Par” and “L-Par-ve”

Table 5.2 shows that for Project “L-Parameter”, the best fit to the measured conductivity data arose using a slightly lower saturated hydraulic conductivity and a tortuosity parameters of 2.4×10^{-6} . For Project “L-Par-ve”, the best fit was obtained with a tortuosity parameter of -0.137 . Figure 5.13 shows the fit to the experimental data; the negative tortuosity parameter produced a better match to the observed conductivity at suction=54.87 cm. The water content data show that the integrity of the previous optimisation was preserved (i.e., the new conductivity data did not disrupt the water content predictions).

5.3 Effects of Model’s Initial Hydraulic Parameters on Inverse Solution

In this section a series of simulations will demonstrate how the starting hydraulic parameters can affect the final solution and run time. The results obtained from the six simulations are compared with those obtained from run ‘a3’. In the following simulation, we only vary parameters Alpha and n; the other parameters are fixed at their optimum value. Details of the six simulations are given in Table 5.3.

Table 5.3: Effect of initial parameters on inverse runs

	Initial parameters		Simulation type	Quality of results	Run time (seconds)
	Alpha	n			
Sen 1	0.0015	1.5	Forward	Very bad	1
Sen 2	0.0015	1.5	Inverse	Very bad	12
Sen 3	0.008	2	Forward	Fair	1
Sen 4	0.008	2	Inverse	Very good	24
Sen 5	0.02	2	Forward	Good	1
Sen 6	0.02	2	Inverse	Very good	14

1. Simulation ‘Sen-1’: We start with arbitrary estimates of Alpha and n. Figure-5.14 shows erroneous results that do not match the observed values.
2. Simulation ‘Sen-2’: Here we run Project ‘Sen-1’ in inverse mode to produce a poor estimate for the two optimised parameters.

3. Simulation ‘Sen-3’: Based on concepts outlined in Appendix I.3, we ran a few trial forward runs and came up with a refined estimate for the initial values for Alpha and n. Figure 5.14 shows that the results now at least are within the range of observed values.
4. Simulation ‘Sen-4’: Use the initial parameters of Project ‘Sen-3’ and run in inverse mode to refine the parameters. The final solution produces results identical to those obtained with run ‘a3’ (Sen-4 is hence not included in Figure 5.14).
5. Simulation ‘Sen-5’: Further manual refinement of the parameters results in a closer fit than that obtained from run ‘a3’.
6. Simulation ‘Sen-6’: Use the initial parameters of Project ‘Sen-5’ and run the inverse solution to refine the parameters. We obtain the same results as from simulation ‘Sen-4’, so why going through the additional effort? The run time is reduced by only 10 seconds! Is this worth it? Yes, if we think in relative terms; 40% less. For complex 2-D runs, the run time could amount to days and such a saving in run time may be significant.

5.4 Long Inverse Trials With a Multiple-Layer Soil Profile

In this section we demonstrate further complications introduced as a result of having a multiple-layer soil profile. The data set was obtained from the Department of Land & Water Conservation, Center of Natural Resources, Soils & Salinity Unit, NSW, Australia (A. Rancic; personal communications).

Simulation ‘LongRun’ shows such a scenario. A 2-m, 3-layered soil system is modelled (layer interfaces are at 30 cm and 110 cm). The lower BC is freely drained and the surface is open to the atmosphere. A 2-year climate data series with transpiration is simulated (root water uptake is modelled). We have water content calibration data (such as neutron probe data) at three depths; 10, 50, and 140cm.

5.4.1 Sensitivity Analysis:

In addition to concepts related to the sensitivity analysis in Appendix I.3, we will highlight several points here: the effect of a nearby boundary, and the interaction of parameters from various layers. The input parameters for all simulations are listed in Table 5.4. The original parameters belong to Project “LongRun”; the three variations for the sensitivity analysis are marked Runs A, B, and C (they are not included on the CD).

1. The effect of a nearby boundary: Refer to Figure 5.15 (Run A). Doubling the hydraulic conductivity (from 50 to 100) of the surface layer had very little effect on the water content predictions at 10 cm.

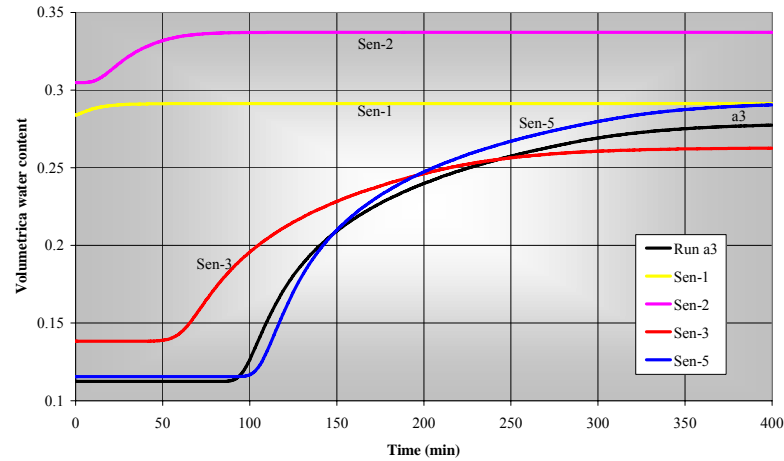


Figure 5.14: Effect of initial parameter values on inverse solution

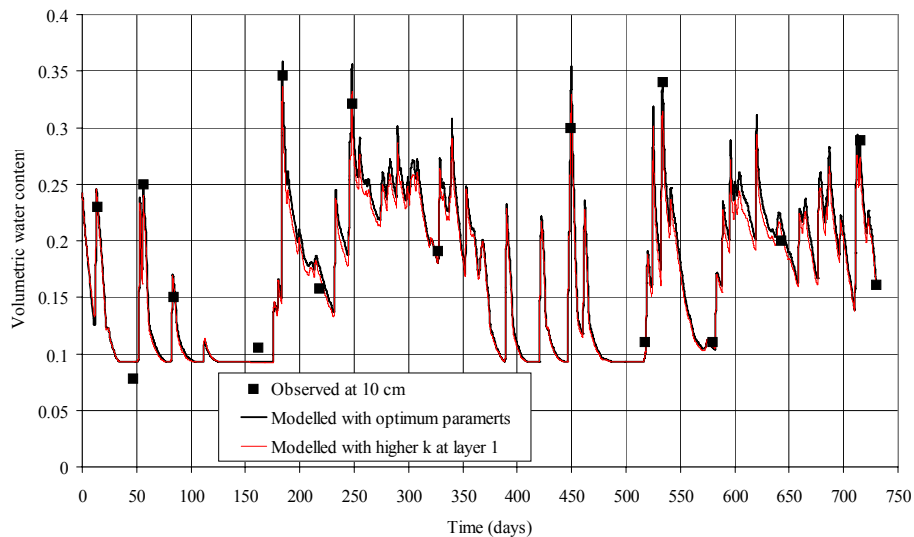


Figure 5.15: Run A (10 cm data), higher Ks for layer 1

Table 5.4: Input parameters for simulations 'LongRun'

	Layer 1; 0-30 cm			Layer 2; 30-110 cm			Layer 3; 110-200 cm		
	Alpha	n	k	Alpha	n	k	Alpha	n	k
LongRun	0.036	1.56	50	0.02	1.41	10.8	0.036	1.56	25
Run A	0.036		100						25
Run B	0.036		50						50
Run C	0.01		50						25

However, Figure 5.16 shows that doubling the hydraulic conductivity (from 25 to 50) of layer 3 (Run B) had a much more significant effect on the simulation results at 160 cm. This is attributable to the fact that this layer is freely drained. Subtle changes in the hydraulic conductivity can have a direct impact on the amount of water exiting the system, and hence would affect the simulation results

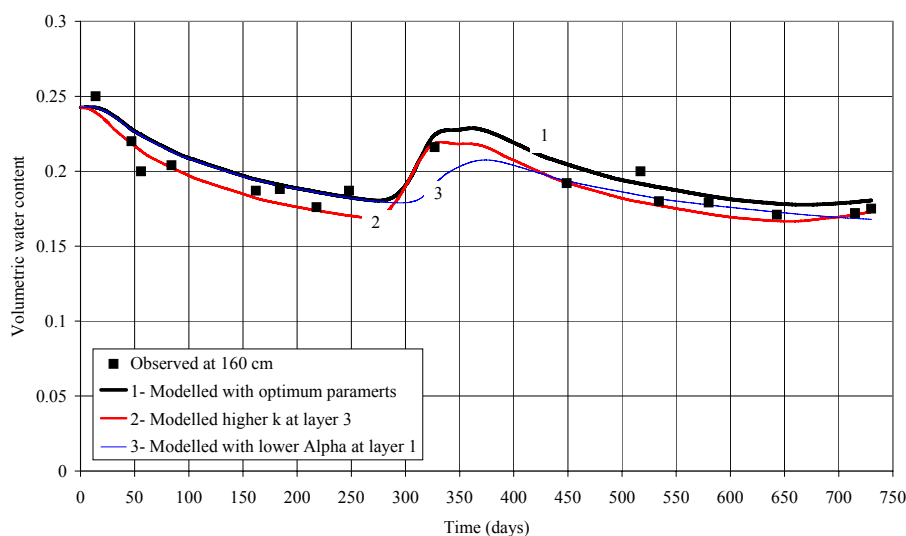


Figure 5.16: Runs B and C (160 cm data), higher K_s for layer 3, and lower Alpha for layer 1, respectively

2. **Parameter interaction:** Figures 5.16 and 5.17 show that changing Alpha for the 1st layer had a major impact on the simulation results for the 2nd and 3rd layers (depths of 50 and 160 cm, respectively).

5.4.2 Inverse Simulations With Layered System

Simulations presented in Section 5.4.1 are often conducted to evaluate the amount of recharge to the groundwater table from cultivated land (represented by cumulative free drainage). They are highly relevant for studying salinity in arid regions.

Note that simulation ‘LongRun’ has the correct water retention parameters (listed in Table 5.4). This case yields a cumulative free drainage of 7.7 cm^2 ; **this is our reference key prediction from the inverse modelling process.** Predictions from all subsequent simulations will be compared to this value.

Our objective here is not to come up with a magical procedure that yields the correct answer, and is applicable to all situations. The objective is to demonstrate the complexity of the problem, to show how different procedures may lead to different answers, and how to best tackle such a problem.

Projects ‘Long-‘

Simulation ‘Long-10’: We calibrate the parameters of the first layer against the calibration data obtained from the same layer. Parameters for layers 2 and 3 are set at their original values. Table 5.5 shows that the resulting parameters are very close to the original parameters listed in Table 5.4 (with the exception of K). This simulation also produced the best estimate for free drainage (result was identical to that from ‘LongRun’ and is hence not included in Figure 5.18). This is attributable to the fact that only three parameters for the first layer were optimised.

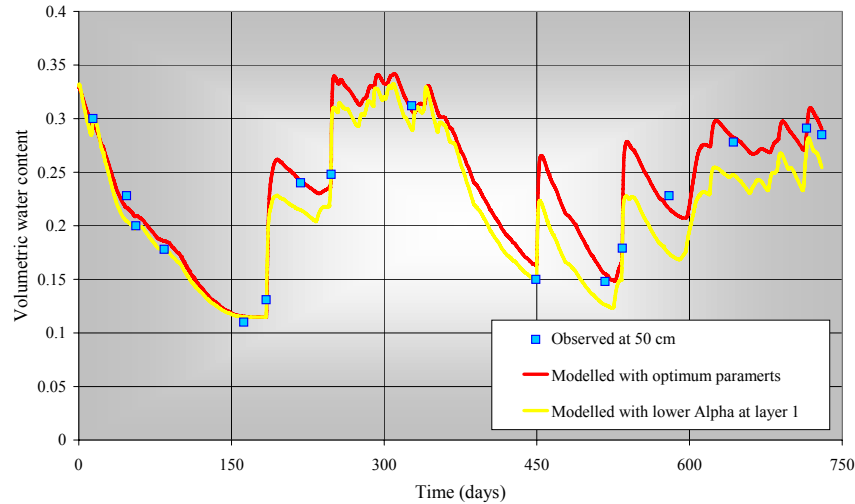


Figure 5.17: Run C (50 cm data), lower Alpha for layer 1

Simulation ‘Long-50’: We calibrate the parameters of the three layers against the calibration data obtained from the second layer. Table 5.5 shows that the optimised parameters for layer 2 are in close agreement to the original. However, the results are different for the other two layers.

Simulation ‘Long-All1’: We calibrate the parameters of the three layers against the calibration data obtained from the three layers. A different set of results is now obtained. Note the high value of the optimised K_s for layer 3; this is a typical case where imposing a maximum limit becomes beneficial and can stop the model from assigning physically unrealistically high values.

Simulation ‘Long-All2’: As above, but use different initial parameters. Yet another set of results is obtained.

Table 5.5 and Figure 5.18 shows that the cumulative free drainage flux is different for the four simulations. However, results obtained from Project ‘LongAll2’ are significantly higher than the rest.

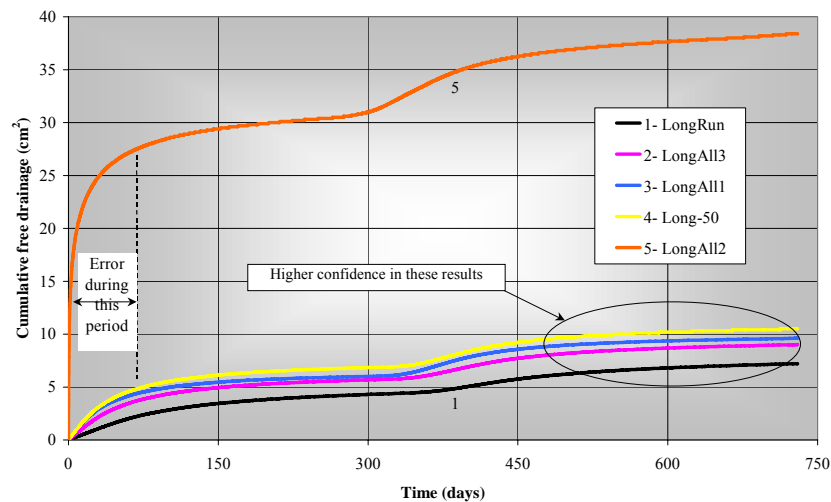


Figure 5.18: Cumulative free drainage flow from the four simulations

Note that although the free drainage prediction obtained from Project ‘LongAll2’ is considered erroneous; the fit to the observed values was excellent (see Figures 5.19-5.21).

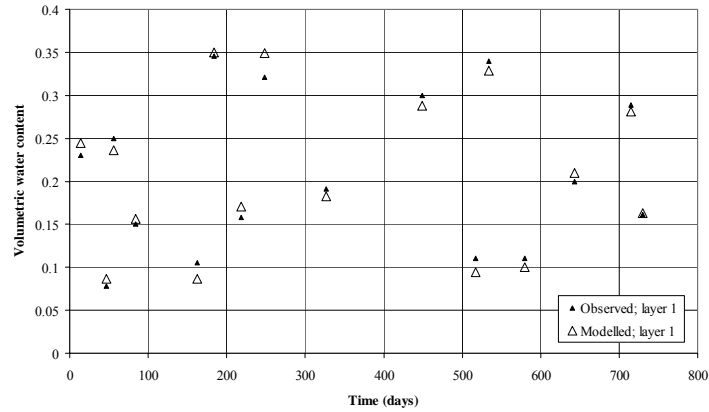


Figure 5.19: Observed and fitted water contents at depth=10 cm; Project ‘LongAll2’

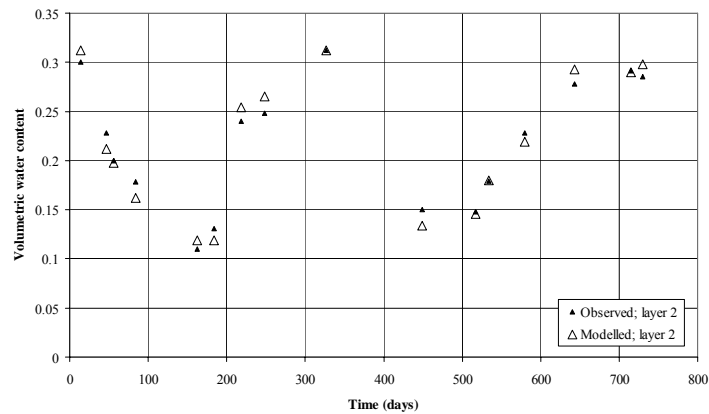


Figure 5.20: Observed and fitted water contents at depth=50 cm; Project ‘LongAll2’

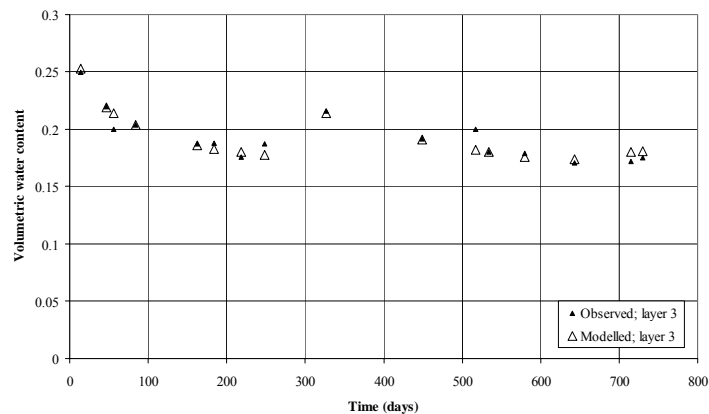


Figure 5.21: Observed and fitted water contents at depth=140 cm; Project ‘LongAll2’

Table 5.5: details for Project series ‘Long-‘

Project name		Long-10			Long-50			Long-All1			Long-All2			Long-All3		
Soil layer		L-1*	L-2	L-3	L-1	L-2	L-3	L-1	L-2	L-3	L-1	L-2	L-3	L-1	L-2	L-3
Input parameters	Alpha	0.02	0.02	0.036	0.02	0.03	0.04	0.02	0.04	0.05	0.001	0.002	0.001	0.001	0.002	0.01
	n	1.4	1.41	1.56	1.5	1.41	1.65	1.4	1.55	1.3	2	1.25	2	2	1.25	2
	Ks	40	10.8	25	30	10.8	40	40	20	100	100	20	25	100	20	25
Optimised parameters	Alpha	0.037	-	-	0.024	0.021	0.034	0.025	0.016	0.073	0.0207	0.0123	0.00198	0.031571	0.01649	0.044808
	n	1.553	-	-	1.68	1.38	1.6	1.68	1.43	1.36	1.726	1.435	1.509	1.6132	1.4558	1.4706
	Ks	79	-	-	22.7	10.7	74.3	36.2	5.5	608	28.35	2.18	189.3	50.589	5.176	100
Calibration data		Data at 10 cm depth			Data at 50 cm depth			Data at depths 10, 50, and 140 cm			Data at depths 10, 50, and 140 cm			Data at depths 10, 50, and 140 cm		
Cumulative free drainage cm ^{2**}		7.28			10.5			9.61			38.4			9.01		

*L refers to Layer

**Reference value = 7.7

What do the results in Figure 5.18 tell us? In real-life situations we generally do not know what the correct answer is, but we may have a good feel for it (know its order; for example, we may think it should be 5 not 50). The comparison in Figure 5.18 indicates a high likelihood that the result obtained from Project ‘LongAll2’ is erroneous, or at least requires further investigation. The fact that three scenarios produced results in the vicinity of 7-10 cm² makes us believe these figures more and disbelieve those obtained from Project ‘LongAll2’. This is another case demonstrating the effects of the initial parameters on the inverse solution, and the crucial role of the modeller’s personal judgement.

Figure 5.22 shows the source of the error. It occurred mainly at the beginning of the simulation; notice the high water content predictions for times <14 (prior to the first calibration data). The prediction at time=14 is excellent (observed=0.3; fitted=0.312); hence, plotting observed versus fitted points only at observation times as shown in Figure 5.21 might mislead the modeller (data from “Fit.out” output file). This discrepancy for times <14 days could very well be overlooked. HYDRUS provides comparisons of fitted data with observation node data, which gives a more complete picture.

Let us revisit the “Variable Boundary Conditions” and look at the precipitation data. We see that the rainfall up to time =14 does not justify the high free drainage shown in Figure 5.18. A close look at the water content data shows that the water content is 0.42 at time=0. If we go back to the sensitivity analysis performed in Appendix I.3, we see that an overestimation of the initial water content is caused by a low Alpha (you can also see this by simply substituting values in the van Genuchten model).

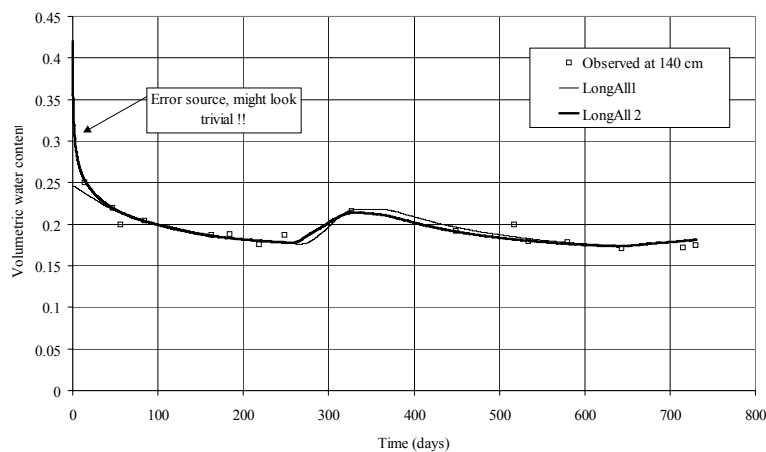


Figure 5.22: comparison of observed and predicted water contents for two simulations

Simulation ‘Long-All3’: We will attempt to solve the problem in Project ‘LongAll2’ by following two steps:

1. Increase the initial Alpha for layer 3 to 0.01. This will result in an initial water content of 0.327 at depth =140 cm (substitute initial values in the van Genuchten model).
2. Introduce minimum and maximum limits to the parameter values to avoid unrealistic estimates obtained in the previous runs (very high K values for

layers 2 and 3). Note that the model failed to run without these limits (the model crashed).

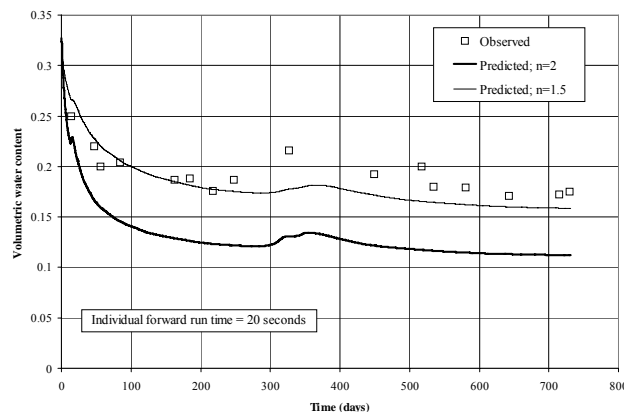


Figure 5.23: Comparison of results from two forward simulations

3. We can investigate the effect of increasing Alpha to 0.01 by running a forward simulation (under ‘Inverse Solution’, set maximum number of iterations to zero). The result is shown in Figure 5.23 (the forward run took about 12 seconds). Notice that the initial water content is 0.32. The entire curve seems to be shifted downward. We demonstrated in Appendix I.3 that the curve could be translated upwards by reducing the parameter n. Use $n=1.5$ and re-run. Figure 5.23 shows that the prediction is now very close to the observed values. We will keep the initial $n=2$ and run simulation ‘Long-All3’ in inverse mode.

Figure 5.18 shows that the cumulative free drainage predictions are reasonable and in agreement with the other runs. The final “optimum” parameters are listed in Table 5.5.

Concluding Comments

Users should be wary of the complications associated with inverse simulations. Close agreement between predicted and observed data does not necessarily mean that the correct answer is obtained.

One should investigate different scenarios, especially for relatively complex problems involving many unknown parameters. Such scenarios include different initial parameters, calibrating various soil layers against the different calibration data, and perhaps excluding results that deviate significantly from other scenarios.

For a multi-layered soil system, a parameter belonging to one layer may affect the predictions for another layer.

Setting minimum and maximum limits for the parameter limits avoids the model from assigning physically unrealistic values, which is advantageous in two ways; firstly, it limits the search domain, thus reducing both run time and the likelihood of obtaining erroneous results, and secondly, reduces the risk of crashing as a result of numerical instabilities that are likely to occur when extremely low or high parameter values are

assigned. It is always a good idea to specify the minimum for the n parameter ($n > 1.05$) when using the van Genuchten's model.

The model may be more sensitive to parameters belonging to the soil located next to an outer boundary such as a free drainage BC or a seepage face.

5.5 General Rules; Dos and Don'ts in Inverse Modelling

1. Having the correct conceptual model is always a crucial element of good modelling (e.g., correct BCs). When conducting an inverse analysis, sometimes a completely insensible result may indicate inadequate conceptualisation of the problem.
2. Even if the model fitted the experimental data nicely, that does not necessarily mean that the correct result has been achieved. Some degree of scepticism is always useful.
3. Limit the number of optimised parameters as much as possible, especially if there is a high correlation between the optimised parameters. The number of optimised parameters should always be equal or less than the number of calibration points. This is like solving a set of linear equations (each calibration point represents an equation): the number of unknowns (optimised parameters) that you can estimate must be equal to the number of equations. For example, if you have four calibration points and five unknown parameters, there will be an infinite number of solutions for such a problem.
4. It is highly recommended to have a second type of calibration data; even one data point could make a tremendous difference. For example, if the calibration data set comprises transient pressure heads, a final water content measurement would be extremely useful.
5. If no good initial estimates for the model parameters are available, try running the model first a few times in forward mode to get a feel for the final results. This should produce a good set of initial parameters in addition to being able to constrain minimum and maximum values. The procedure will be time saving in the end, especially when running long simulations.
6. Even if a good set of starting parameters was selected, try other values to make sure that the model converges to the same solution.
7. Define the final objective (the key prediction) of the modelling process. For example, let's assume that the goal is to estimate the cumulative flow across a boundary, like seepage from a dam. Identify first which parameter the flow is most sensitive to. For a multi-layered soil system, common sense suggests that parameters belonging to the soil next to the seepage boundary is the most crucial; hence, there should be a good calibration data set for that area. Another example involves high-intensity rainfall areas where the saturated conductivity of the surface soil will determine how much water will infiltrate into the soil; this parameter hence is likely the most important.
8. Make sure that your calibration data are evenly distributed spatially and temporally to eliminate any bias in the results. For example, if you are trying to evaluate the water retention curve of a soil, your calibration data should cover water contents from saturation down to points where the retention curve changes its slope (where it starts curving toward the residual value). High data

density (at some place or during some time period) results in skewing the calibration.

9. Highly correlated parameters often lead to non-unique solutions. That is, more than one set of parameters will result in a similar fit to the calibration data. In these cases, either eliminate one of the correlated parameters from the optimisation procedure (if you have a reasonably good independent estimate for that parameter), or introduce new types of calibration data. For example, if your calibration data consist of pressure heads in the saturated zone and the K_s values of a draining two-layered soil system are highly correlated (model responds to K_1/K_2 ratio), you probably should try to measure the flow rate. The more complex a model, the more likely non-uniqueness will become a problem.
10. Weighing factors are useful, but do not use them unless you are very confident of the data point to which you are assigning a high weight. For example, see the ‘Crust’ example included in the HYDRUS package; a single measurement on the retention curve was given a weight of 10. You can also assign higher weights to data obtained from areas where predictions are important (e.g. data next to a boundary where flow needs to be predicted).
11. Since inverse runs are generally time-consuming, it is always worthwhile to first invest some time in devising a mesh that runs as fast and accurately as possible. That is, use the coarsest possible mesh and loosen tolerances (while still getting a reasonable mass balance). Once you come up with a good preliminary result, tighten the tolerances to obtain the final answer.
12. Even if you obtain a nice fit between observed and modelled data, always closely investigate “Inverse Solution” and “Mass Balance” under Post-processing. Look at the statistical information provided and make sure that the results are physically realistic.
13. Parameters with very wide confidence limits may be eliminated from the optimisation process since the model is not sensitive to them.
14. Validation is always another way of confirming that you obtained the correct results. You must validate against data that were not used in the calibration process.
15. **And finally: remember that the inverse method is not the black box approach that always gives the right answer; some subjective judgement of the modeller may well be critical in many problems.**

Final remark: We refer readers to Section 1.6, 3.4.2, and 3.6 of the “**Method of Soil Analyses**” book of the Soil Science Society of America for a detailed discussion of the theory behind parameter estimation technique, and various applications for estimating soil hydraulic and solute transport parameters (Šimůnek and Hopmans, 2002; Hopmans et al., 2002; and Šimůnek et al., 2002).

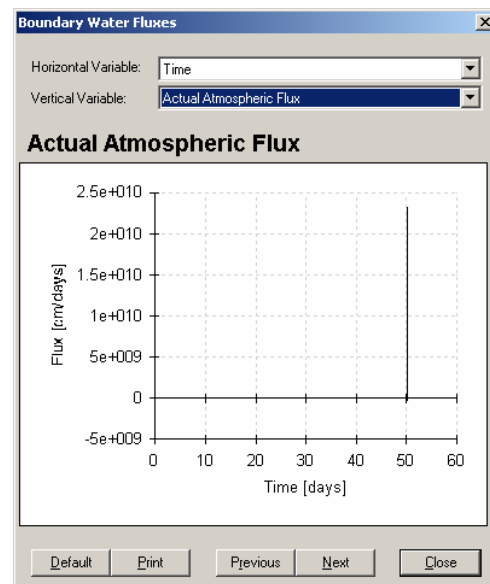
6. TROUBLE SHOOTING

6.1 Infiltration After Long Dry Period

Project Mesh-1:

This project simulates infiltration into a silty soil after a long evaporation period. Project 'Mesh1' is run using the default time stepping parameters. The results are shown below. Projects 'Mesh 1 & 2' are found on the CD (..\HYDRUS Manual Projects\Sect-6).

Checking the actual atmospheric flux reveals an error at time=50 days; notice the magnitude of the flux (10^{10} cm is equal to 500,000 kilometres!!).



Checking the mass balance information confirms the presence of serious errors between times 50 and 51 days

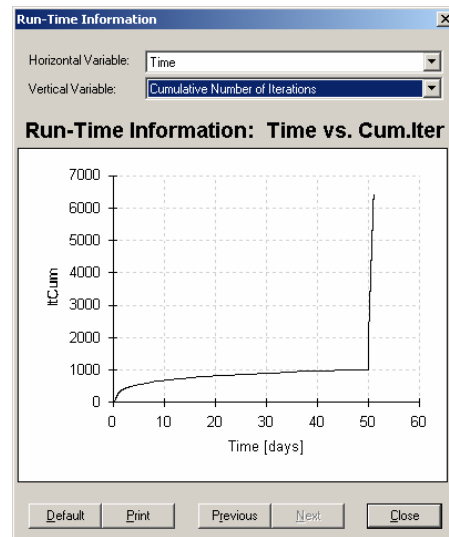
The figure is a screenshot of a software window titled "Mass Balance Information". It displays a table of mass balance data. The table is divided into two sections by dashed lines. The first section shows data for the entire simulation, and the second section shows data for sub-region 1 at time 51.0000. Red dashed boxes highlight significant errors in the 'hMean [L]', 'WatBalT [V]', and 'WatBalR [%]' rows for the second section.

Variable	Value 1	Value 2
Volume [V]	.29576E+03	.29576E+03
InFlow [V/T]	-.82470E+00	-.82470E+00
hMean [L]	-.99695E+03	-996.954
WatBalT [V]	.42328E-01	
WatBalR [%]	-.026	

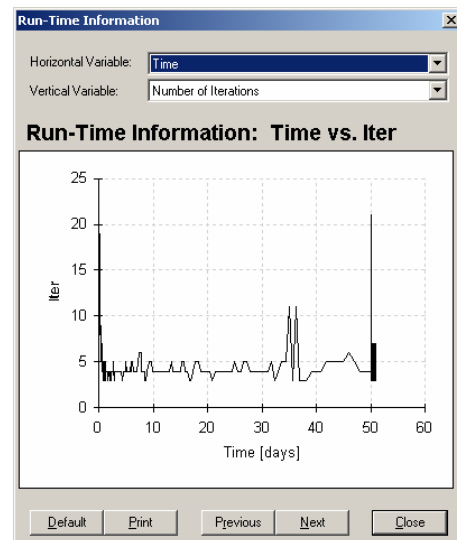
Time [T]	51.0000	

Sub-region num.	1	
Area [V]	.10000E+04	.10000E+04
Volume [V]	.46000E+03	.46000E+03
InFlow [V/T]	.00000E+00	.00000E+00
hMean [L]	804.48E+00	.804
WatBalT [V]	.32748E+09	
WatBalR [%]	71.377	

The sudden increase in the number of iterations at around time = 50 days is also suspicious and should be investigated, even though it may not necessarily indicate an error.



This figure suggests that the solution did not converge at time=50 days (number of iterations exceeded the maximum limit of 20).

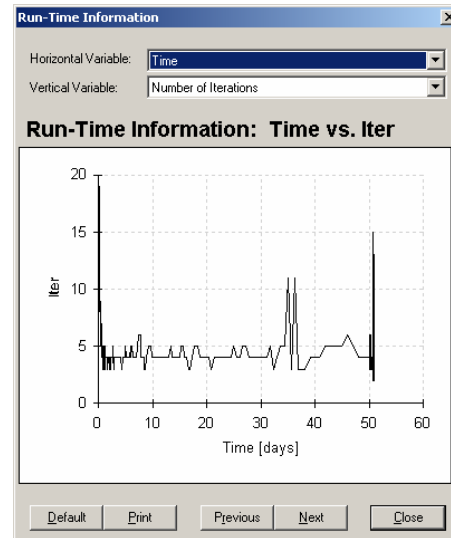
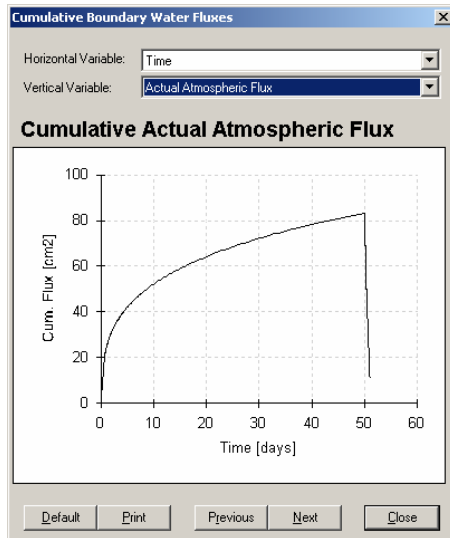


Spreadsheet 'Mesh1.xls' shows the output file 'h-mean.out'; notice the errors at time=50 day (shown below). If you try to access the pressure head data in HYDRUS, you will get an error message; this is due to the errors shown below in 'h-mean.out'

50.002	-988063	0	0	0	0	0	-138.3
50.003	59811490	0	0	0	0	0	-138.3
50.0040	*****	0	0	0	0	-138.3	
50.0050	*****	0	0	0	0	-138.3	
50.0060	*****	0	0	0	0	-138.3	
50.0070	*****	0	0	0	0	-138.3	

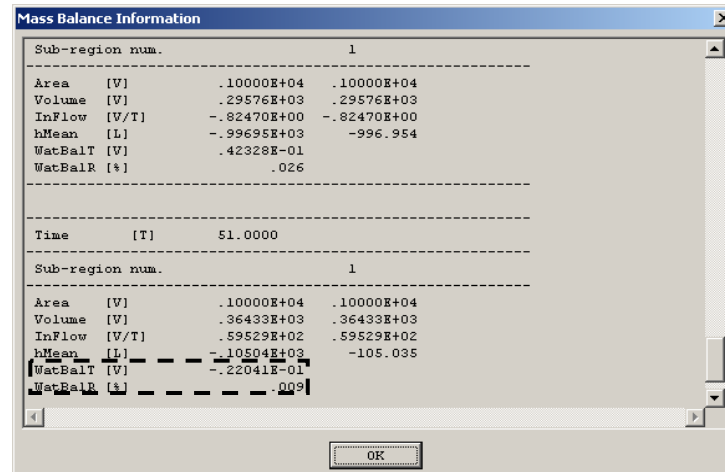
Project Mesh-2:

The problem encountered in Project ‘Mesh1’ is fixed by lowering the minimum time step to $1E-5$ day. This will allow the model to lower the time step to a small enough value that will accommodate the numerical instabilities encountered when precipitation is suddenly introduced at time=50 days.



Actual atmospheric flux is now correct.

There is now no divergence in the solution at time=50 days.



The mass balance results are now acceptable.

Conclusion: Always allow small time steps by specifying a “Minimum Time Step” on the order of seconds.

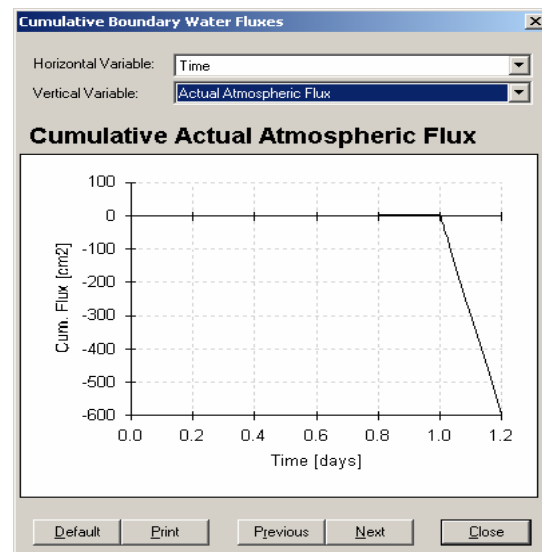
6.2 High Infiltration Rate Into a Two-Layer Soil System

Project Mesh-3:

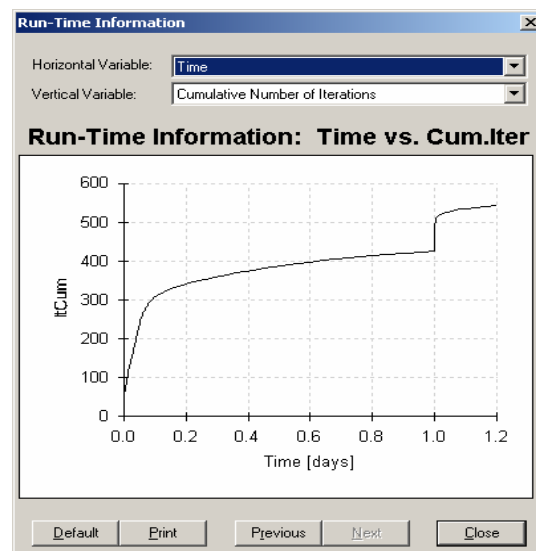
This project simulates infiltration into a two-layered coarse-above-fine soil system. The soil profile is initially relatively dry (uniform pressure head=-100 cm). The profile further dried out due to evaporation for 1 day. An intense rainstorm then occurred at a rate of 300 cm/day. Project 'Mesh3' is run using the default time stepping parameters. The results are shown below. Projects 'Mesh 3 & 4' are found on the CD (.\HYDRUS Manual Projects\Sect-6).

Checking the actual atmospheric flux does not reveal a problem. The magnitude of the flux is equal to the potential rate ($0.2 \times 300 \times 10 = 600$), which is $\text{time} \times \text{intensity} \times \text{width}$.

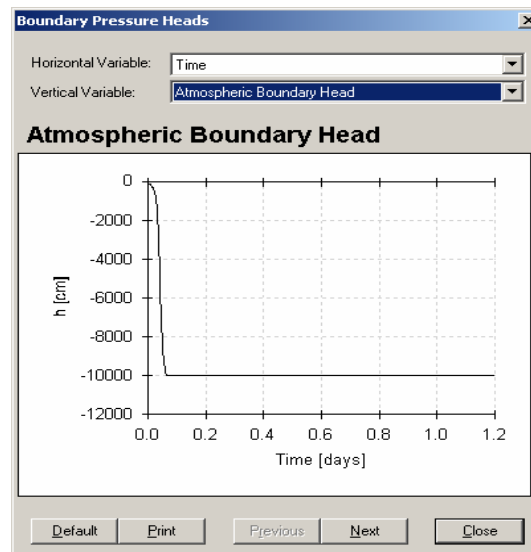
This value could be believable if you overlook the fact that the introduced flux exceeds the saturated conductivity of the surface layer.



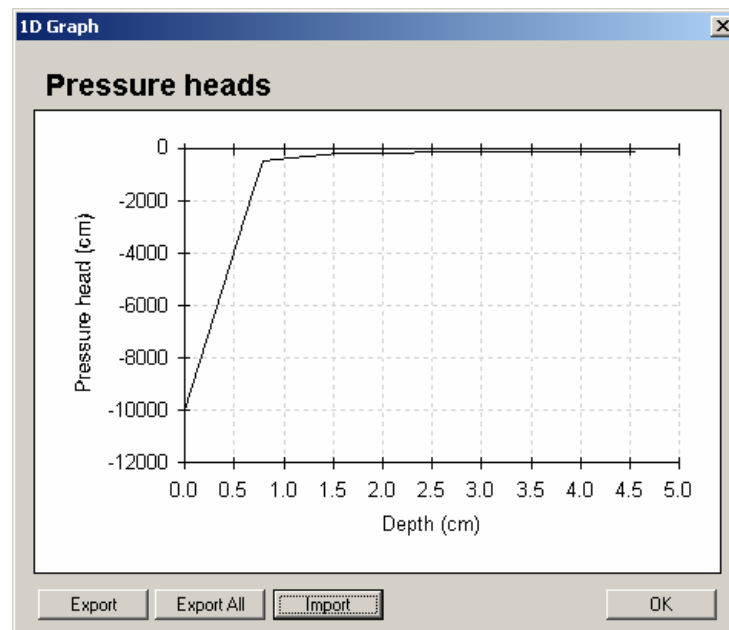
The rapid increase in the cumulative number of iterations at time=1 day is expected but should be investigated.



The atmospheric boundary pressure head drops to h_{CritA} (-10,000) due to evaporation, which is correct. However, the pressure head does not increase due to rainfall at $\text{time}=1$. Why??



A graphical display of the results reveals that the soil surface at the end of the simulation is completely dry; No rainfall was apparently introduced.



Checking the mass balance at time >1 day reveals extremely large errors.

Mass Balance Information			
Sub-region num.	1	2	
Area [V]	.10000E+04	.39806E+03	.60194E+03
Volume [V]	.17940E+03	.47712E+02	.13168E+03
InFlow [V/T]	-.49288E+00	-.45462E+00	-.38259E-01
hMean [L]	-.14007E+03	-200.468	-100.129
WatBalT [V]	-.20189E-02		
WatBalR [%]	.153		

Time [T]	1.0800		

Sub-region num.	1	2	
Area [V]	.10000E+04	.39806E+03	.60194E+03
Volume [V]	.17934E+03	.47660E+02	.13168E+03
InFlow [V/T]	-.45456E+00	-.41949E+00	-.35076E-01
hMean [L]	-.14031E+03	-201.037	-100.144
WatBalT [V]	-.24003E+03		
WatBalR [%]	99.453		

Project Mesh-4:

The problem encountered in example Mesh3 is solved here using a simple trick. The variable boundary condition is discretized into 3 stages. A very short (0.001 day, less than 1.5 minute) imaginary period of moderate rainfall (5 cm/day) is introduced before the intensive-rainfall period. The problem is solved.

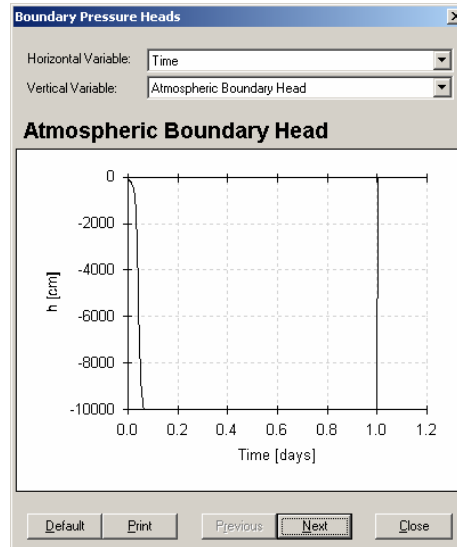
Mass balance is restored.

Mass Balance Information			
Volume [V]	.27756E+03	.14588E+03	.13169E+03
InFlow [V/T]	.10603E+04	.10538E+04	.65335E+01
hMean [L]	-.65494E+02	-13.127	-100.125
WatBalT [V]	-.33401E-01		
WatBalR [%]	.034		

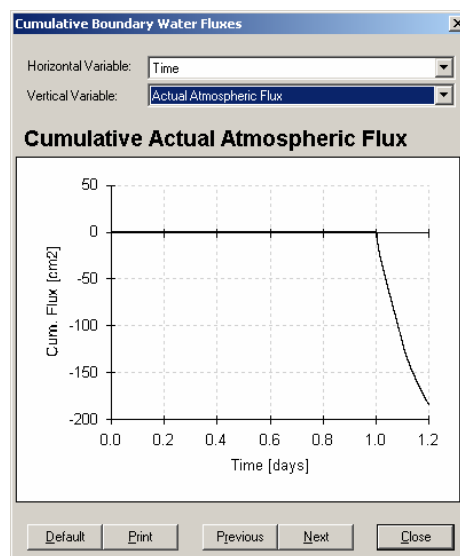
Time [T]	1.2000		

Sub-region num.	1	2	
Area [V]	.10000E+04	.39806E+03	.60194E+03
Volume [V]	.36494E+03	.16313E+03	.20180E+03
InFlow [V/T]	.49642E+03	.00000E+00	.49642E+03
hMean [L]	-.10495E+02	10.314	-24.256
WatBalT [V]	-.13626E+00		
WatBalR [%]	.073		

The pressure head at the soil surface increases to zero when rainfall starts.



The cumulative flux is now much lower than the potential, which is believable since the rainfall intensity is much higher than the saturated hydraulic conductivity of the soil.



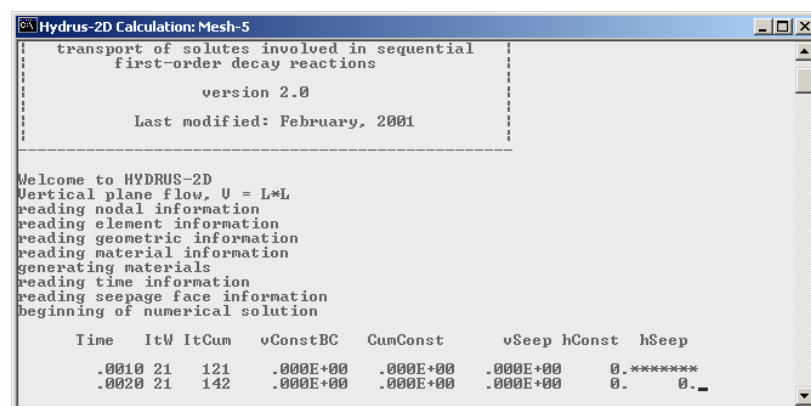
Remark: This problem could have been solved also by allowing smaller time steps, i.e., a “Minimum Time Step”=1e-6 d.

6.3 Simulations With Potentially Unstable Initial Conditions; Drainage of a Saturated Coarse-Grained Soil

Project Mesh-5

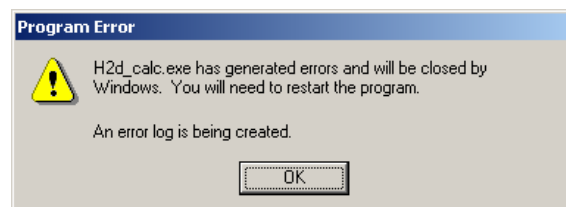
This example demonstrates a case where the initial conditions are potentially unstable, which could lead to divergence problems. The instability is related to the presence of a fully saturated coarse-textured soil profile with a free-draining base and a seepage face along the side. We proceed with setting up the problem and accepting all default values that HYDRUS provides (initial time step=0.1 day; minimum time step=0.001 day; maximum number of iterations=20). Projects 'Mesh 5, 6 & 7' are found on the CD (..\HYDRUS Manual Projects\Sect-6).

The results are shown here. During the first time steps the maximum number of iterations is exceeded.



Time	ItW	ItCum	vConstBC	CunConst	vSeep	hConst	hSeep
.0010	21	121	.000E+00	.000E+00	.000E+00	0.*****	
.0020	21	142	.000E+00	.000E+00	.000E+00	0.	0.

The solution diverges and the program is terminated.

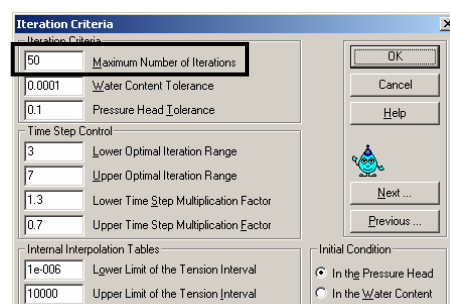
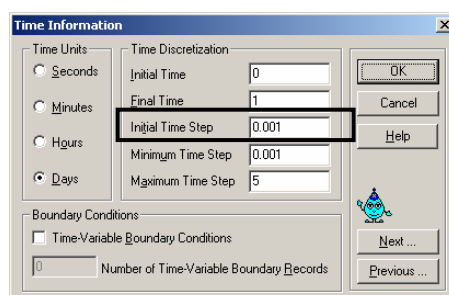


Project Mesh-6

Trying to solve the problem of Mesh-5 by increasing the maximum number of iterations does not work.

Reducing the initial time step to 0.001 day does not work either.

Try doing both. Increase the maximum number of iterations to 50 and reduce the initial time step to 0.001 day.



The solution converges and the problem is solved. Note that the number of iterations during the first time step is 31, higher than the default value of 20.

```

Hydrus-2D Calculation: Mesh-6
-----
first-order decay reactions
version 2.0
Last modified: February, 2001

Welcome to HYDRUS-2D
Vertical plane flow, U = L*L
reading nodal information
reading element information
reading geometric information
reading material information
generating materials
reading time information
reading seepage face information
beginning of numerical solution

Time  ItW  ItCum  vConstBC  CumConst  vSeep  hConst  hSeep
.0010  31   31     .000E+00  .000E+00  .000E+00  0.     -1.
.0020  10   41     .000E+00  .000E+00  .000E+00  0.     -1.
.0030  13   54     .000E+00  .000E+00  .000E+00  0.     -1.
.0040  10   64     .000E+00  .000E+00  .000E+00  0.     -2.

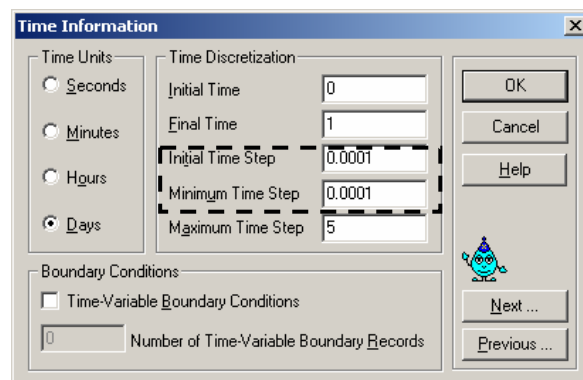
```

Project Mesh-7

Alternatively, solve the problem of Project 'Mesh-5' by reducing the initial time step to 0.0001 day (better option).

Changing the initial time step to 0.0001 day should solve this problem. Make sure that the minimum time step is simultaneously reduced to a value \leq to the initial time step.

The maximum number of iterations should be reset to 20 (to see if reducing the initial time step would solve the divergence problem).



```

Hydrus-2D Calculation: Mesh-7
-----
Welcome to HYDRUS-2D
Vertical plane flow, U = L*L
reading nodal information
reading element information
reading geometric information
reading material information
generating materials
reading time information
reading seepage face information
beginning of numerical solution

Time  ItW  ItCum  vConstBC  CumConst  vSeep  hConst  hSeep
.100000E-03  15   15     .000E+00  .000E+00  .000E+00  0.     0.
.0002  6   21     .000E+00  .000E+00  .000E+00  0.     0.
.0003  4   25     .000E+00  .000E+00  .000E+00  0.     0.
.0004  3   28     .000E+00  .000E+00  .000E+00  0.     0.
.0005  3   31     .000E+00  .000E+00  .000E+00  0.     0.
.0007  4   35     .000E+00  .000E+00  .000E+00  0.     0.
.0009  4   39     .000E+00  .000E+00  .000E+00  0.     -1.
.0010  3   42     .000E+00  .000E+00  .000E+00  0.     -1.
.0013  3   45     .000E+00  .000E+00  .000E+00  0.     -1.
.0015  4   49     .000E+00  .000E+00  .000E+00  0.     -1.

```

The problem is solved!

Conclusion: Allowing smaller time steps is almost always a good idea!

6.4 Infiltration Into a Thick Clay Soil Layer After a Dry Period

Project Mesh-8:

This project simulates a drying and a wetting cycle in a 100-cm thick fine-textured (clay) soil block. The soil is at a uniform initial pressure head of -100 cm. Evaporation is simulated for a period of 0.2 days followed by infiltration up to 1 day. The precipitation rate is equal to the infiltration capacity of the soil (equal to the saturated hydraulic conductivity of 4.8 cm/day). Boundary conditions are atmospheric at the surface and free drainage at the base.

Note that for the first two cases below (1 and 2) we used the optimum time discretisation and iteration criteria (of run Mesh8e). This to be able to isolate the effects of each problem, that is, the high precipitation rate (1), and the soil hydraulic model (2). All related projects are found on the CD (**.\HYDRUS Manual Projects\Sect-6**).

Project Mesh8: use this project to try (1 and 2)

Problems that are likely to be encountered in such a simulation:

1. Precipitation \geq saturated hydraulic conductivity (4.8 cm/day)

Make sure you check the 'with air entry value of -2 cm' box in the soil hydraulic model. Under variable BC, set the precipitation rate to 4.8 cm/day, and run. Notice how the simulation does not progress beyond time > 0.467 day.

Time	ItW	ItCum	CumAtmBC	CumConst	CumDrain	hAtm	hConst	hDrain
.4670	21	81624	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81628	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81649	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81653	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81674	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81678	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81699	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81703	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81724	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81728	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81749	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81753	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81774	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81778	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81799	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81803	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81824	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81828	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81849	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81853	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.
.4670	21	81874	-.205E+02	.000E+00	.195E+01	-982.	0.	-100.
.4670	4	81878	-.205E+02	.000E+00	.195E+01	-2.	0.	-100.

Solution: decrease the precipitation slightly to 4.7 cm/day and the simulation will run in only 48 seconds with a total number of iterations of 16,239.

2. Soil hydraulic model used

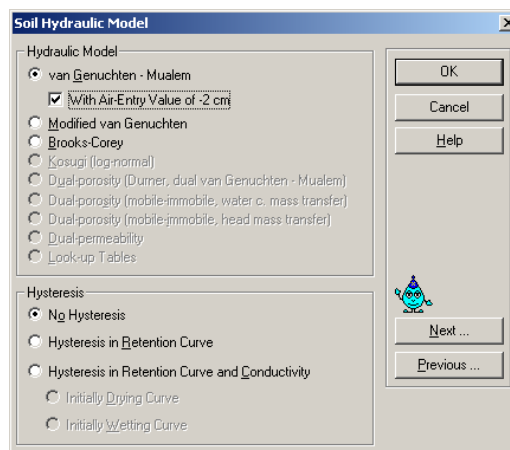
The preferred model for very fine-textured (clayey) soil is the van Genuchten-Mualem model with an air entry value of -2 cm. This model improves numerical stability for a soil that has a highly nonlinear hydraulic conductivity function close to saturation.

Reset the precipitation rate to 4.7 cm/day in **Project Mesh8**, and uncheck the 'with air entry value of -2 cm' box in soil hydraulic model. Run the model.

The screen capture below shows the progress of the simulation when the normal van Genuchten-Mualem model is used. Notice the number of iterations and the time. The simulation almost does not progress beyond time >0.246 day.

Time	ItW	ItCum	CumAtmBC	CumConst	CumDrain	hAtm	hConst	hDrain
.2460	2124566		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124568		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124570		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	4124594		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	6124600		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124602		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124604		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	3124607		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	11124618		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	3124621		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	3124624		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	4124648		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	6124654		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124656		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124658		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124660		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124662		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	6124668		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124690		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	4124694		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	6124700		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.
.2460	2124702		-.321E+00	.000E+00	-.116E+00	0.	0.	-1.00.

Solution: Select the soil hydraulic model as shown below



3. Time discretization and iteration criteria

For Projects Mesh-8a to Mesh-8e: use the van Genuchten model with a -2 cm air entry value, and a surface flux= 4.7 cm/day.

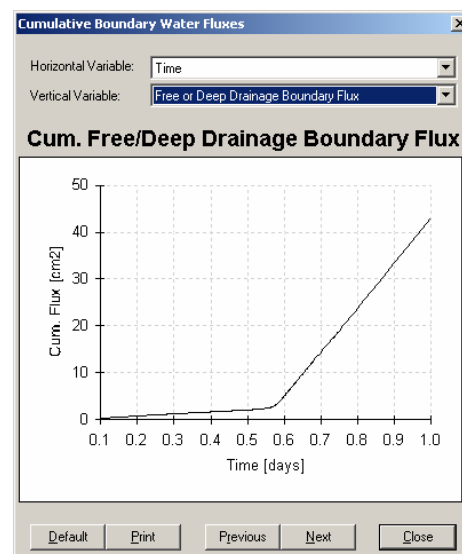
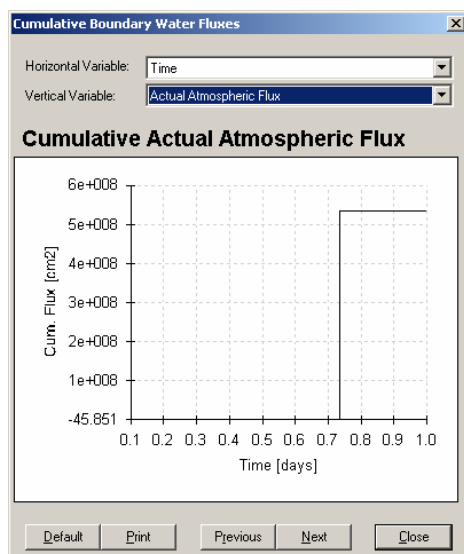
- **Project Mesh-8a:** using default time step and iteration criteria (run time = 11 sec.)

The mass Balance results are very poor

WatBalT [V] 0.53537E+09
WatBalR [%] 100.000

Select Hydrus-2D Calculation: mesh-8a									
.6916	2	1978	-.422E+02	.000E+00	.137E+02	-2.	0.	-2.	
.6958	2	1980	-.426E+02	.000E+00	.141E+02	-2.	0.	-2.	
.7000	2	1982	-.430E+02	.000E+00	.145E+02	-2.	0.	-2.	
.7048	2	1984	-.434E+02	.000E+00	.149E+02	-2.	0.	-2.	
.7111	2	1986	-.440E+02	.000E+00	.155E+02	-2.	0.	-2.	
.7192	3	1989	-.448E+02	.000E+00	.163E+02	-2.	0.	-2.	
.7293	14	2003	-.450E+02	.000E+00	.172E+02	0.	0.	0.	
.7303	21	2064	-.457E+02	.000E+00	.173E+02	*****	0.	*****	
.7313	21	2085	-.435E+02	.000E+00	.174E+02	*****	0.	*****	0.920322.
.7323	21	2106	.298E+05	.000E+00	.174E+02	*****	0.	*****	0.*****
.7333	21	2127	.298E+05	.000E+00	.175E+02	*****	0.	*****	0.*****
Time	ItW	ItCum	CumAtmBC	CumConst	CumDrain	hAtm	hConst	hDrain	
.7343	21	2148	.535E+09	.000E+00	.175E+02	*****	0.	*****	
.7353	21	2169	.535E+09	.000E+00	.176E+02	59.	0.	59.	
.7363	21	2190	.535E+09	.000E+00	.177E+02	11.	0.	11.	
.7373	3	2193	.535E+09	.000E+00	.178E+02	0.	0.	0.	
.7386	3	2196	.535E+09	.000E+00	.179E+02	0.	0.	0.	
.7403	3	2199	.535E+09	.000E+00	.181E+02	0.	0.	0.	
.7425	3	2202	.535E+09	.000E+00	.183E+02	0.	0.	0.	
.7454	3	2205	.535E+09	.000E+00	.186E+02	0.	0.	0.	
.7490	3	2208	.535E+09	.000E+00	.189E+02	0.	0.	0.	
.7536	3	2211	.535E+09	.000E+00	.194E+02	0.	0.	0.	
.7603	3	2214	.535E+09	.000E+00	.200E+02	0.	0.	0.	

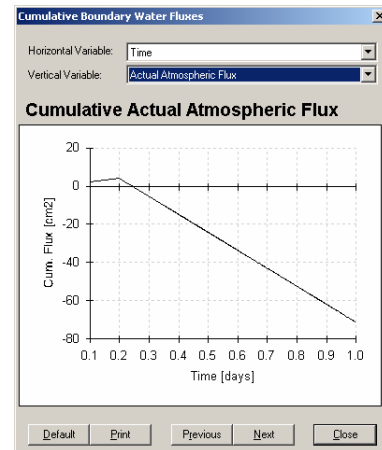
The cumulative actual atmospheric flux is incorrect; HOWEVER, the free drainage flux is correct!!!



- **Project Mesh-8b:** default time step and iteration criteria, but we reduced the number of print times to 1.

The mass balance improved, and the actual atmospheric flux is correct!!

WatBalT [V]	-0.13266E+01
WatBalR [%]	1.100



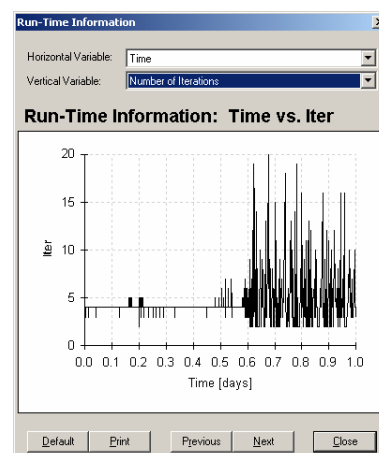
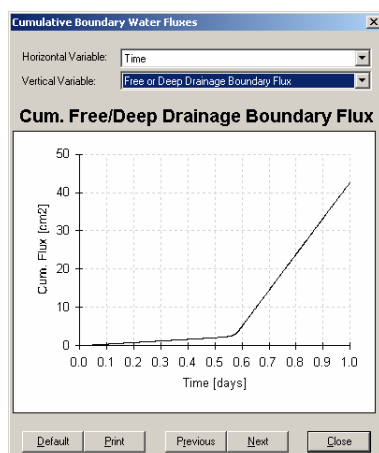
This suggests that in the previous run, print times must have interfered with the time stepping scheme. This is not a solution but an indication that the time stepping parameters were not correct.

- **Project Mesh-8c:** change the minimum time step and the water content tolerance
 - We decrease the initial and minimum time steps to as low as 1E-15 but it does not solve the problem.
 - Try initial and minimum time step = 1E-10 day and tighten the water content tolerance to 1E-7. Run time is 64 sec. And the results are correct. The run time information reveals no divergence problems.

Mass balance:

WatBalT [V]	.45776E-04
WatBalR [%]	.000

Total number of iterations = 22,161



- **Project Mesh-8d:** Modify the lower tension intervals

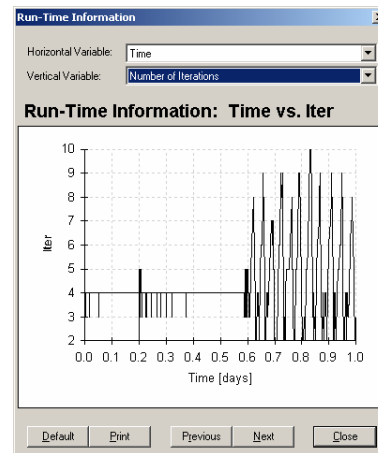
Lower limit of the tension interval = 0.001

WatBalT [V] .75912E-03

WatBalR [%] .001

Number of iterations is well below the maximum.

Total number of iterations = 20,097



- **Project Mesh-8e:** Modify the upper tension intervals

Upper limit of the tension interval = 200

Mass balance:

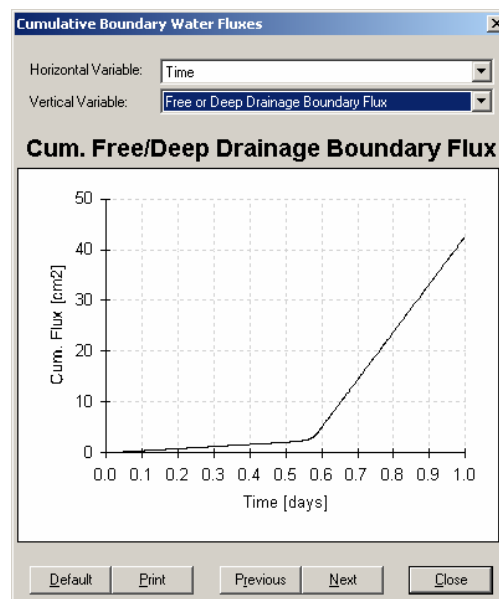
WatBalT [V] .12970E-02

WatBalR [%] .001

Run time = 51 sec.

Total number of iterations = 16,239.

The cumulative free drainage flow is shown here; compare the result with that obtained from project 'Mesh8a' and 'Mesh8c'.



Conclusions: Specified infiltration rates close to the K_s value of heavy textured soils may cause numerical instabilities. We showed above several ways of overcoming these instabilities.

6.5 Simulations With Highly Unstable Initial Conditions; Very High-Pressure Gradient

This series of projects (“Grad-”) demonstrates the difficulties that are likely to be encountered when highly unstable initial conditions are assigned to the domain. The runs simulate infiltration into a 10-cm soil column with extremely high initial pressure gradients of up to 250. The pressure head is assumed to be linearly distributed with depth and the pressure head is set to zero at the free drainage boundary for all simulations. All related projects are found on the CD (..\HYDRUS Manual Projects\Sect-6).

▪ Projects ‘Grad-1’ and ‘Grad-1a’

For simulation ‘Grad1’, the elements were evenly spaced at 1 cm. The pressure head at the surface was set at –1200 cm. Table 6.1 indicates that this grid produced a relatively small mass balance error. Figure 6.1 shows that the number of iterations has exceeded the maximum limit (solution did not converge) during the first 16 time steps.

In Project ‘Grad1a’, we try to reduce the initial time step but this produces extremely poor results! This suggests that the grid should be refined.

Table 6.1 Summary of the results obtained from the seven runs.

Project	Simulation time (day)	Run time (sec)	Surface potential (cm)	Mesh size*	Initial and minimum time step (day)	Mass balance**	Notes on mass balance
Grad1	1	2	-1200	11 1	default [#]	0.13 1.729	fair
Grad 1a	1	1	-1200	11 1	1E-6	1340 99.43	poor
Grad 2	1	2	-1200	51 1	default	0.00466 0.062	good
Grad 3	1	6	-1500	101 0.1	default	0.00028 0.004	good
Grad 4	1	5	-2500	101 0.1	0.0001	1.1E6 52.82	poor
Grad 5	0.1	81	-2500	101 0.1	0.00001 [@]	0.0108 1.798	fair
Grad 6	0.1	144	-2500	201 0.1	0.00001	0.0000348 0.006	good

* Number of elements and density at top of mesh, respectively

** Mass balance volume and % error, respectively

[#] Default values are: Initial time step=0.1; minimum time step=0.001

[@] Reducing the time step to 1E-12 did not improve the results

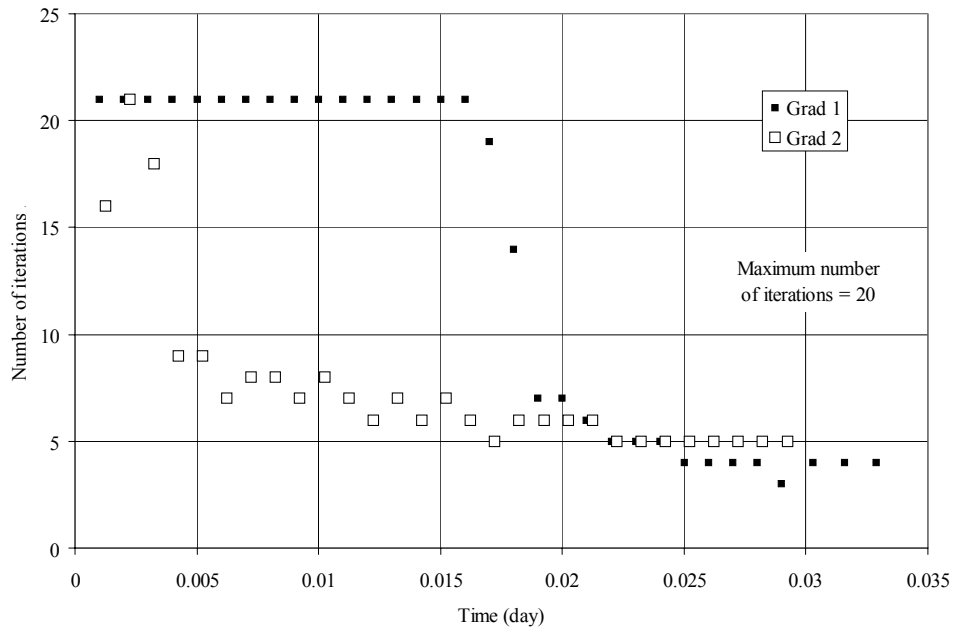


Figure 6.1. Run-time information for projects 'Grad1' and 'Grad2'

▪ Projects 'Grad-2'

For simulation 'Grad2', the grid was refined. The elements were evenly spaced at 0.2 cm (see Table 6.1). The mass balance is now improved. Figure 6.1 shows that the maximum number of iterations was exceeded once only. Figure 6.2 shows that the cumulative free drainage flow was different for the two simulations. The trend obtained for Project 'Grad 1' shows an anomaly at the beginning of the simulation.

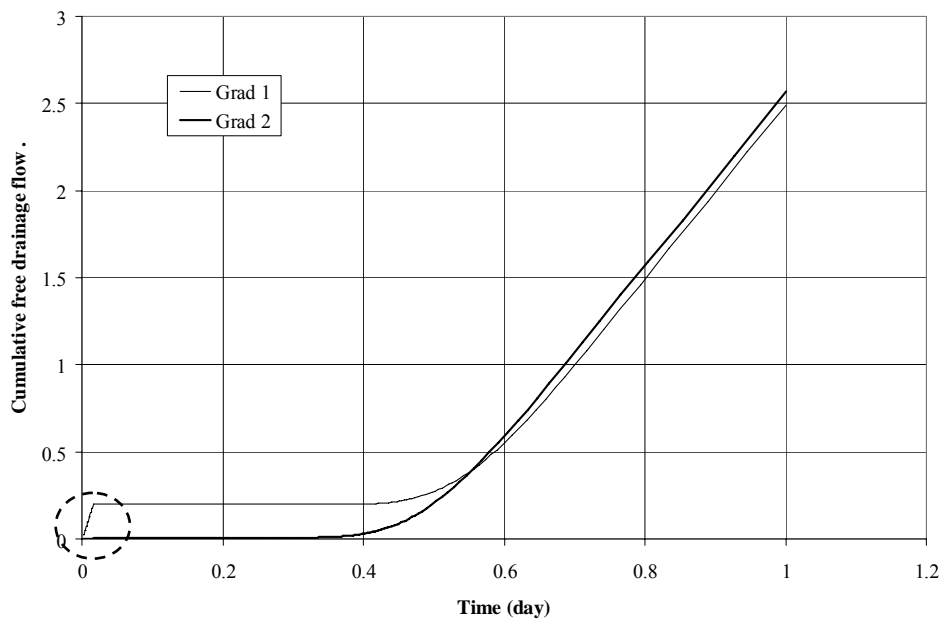


Figure 6.2. Cumulative free-drainage flow for projects 'Grad1' and 'Grad2'

▪ Projects ‘Grad-3’

For simulation ‘Grad3’, the surface pressure head was decreased to -1500 cm. In order to get a stable run with a good mass balance, the grid had to be further refined. The total number of elements was increased to 101 and the density at the top was reduced to 0.1; this resulted in a surface element of 0.02 cm. Results were acceptable (see Table 6.1).

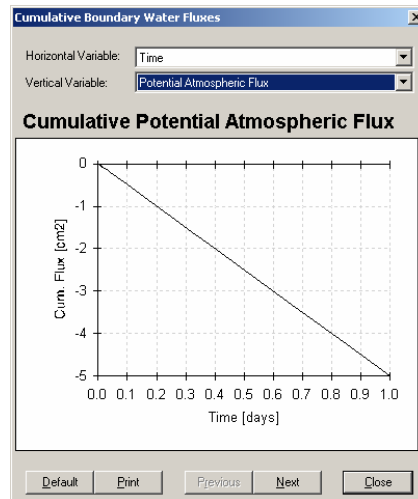
▪ Projects ‘Grad-4’

For simulation ‘Grad4’, the surface pressure head was decreased to -2500 cm. We used the same grid spacing as that for Project ‘Grad3’. In order to get the HYDRUS to run, the initial time step had to be reduced to 0.0001 day. The simulation proceeded as shown below. Notice that the number of iterations always exceeded the maximum limit of 20; also notice that ‘hAtm’ and ‘hDrain’ fluctuate in a chaotic way. Nevertheless, the run proceeds and results are displayed in a normal manner (the run does not terminate automatically as in Project ‘Grad 1’). We will examine the erroneous results. If you encounter such a run, there is no point in continuing it; better terminate it by pressing Ctrl-C.

Time	ItW	ItCum	CumAtmBC	CumConst	CumDrain	hAtm	hConst	hDrain
.0003	21	83	-.201E+04	.000E+00	.501E-02*****	0.	-77.	
.0004	21	104	-.201E+04	.000E+00	.639E-02*****	0.	19.	
.0005	21	125	-.201E+04	.000E+00	.639E-02*****	0.	-80.	
.0006	21	146	-.509E+06	.000E+00	.794E-02*****	0.	36.	
.0007	21	187	-.509E+06	.000E+00	.795E-02*****	0.	-143.	
.0008	21	208	-.509E+06	.000E+00	.104E-01*****	0.	224.	
.0009	21	229	-.509E+06	.000E+00	.104E-01*****	0.	-394.	
.0010	21	270	-.509E+06	.000E+00	.129E-01*****	0.	1768.	
.0011	21	311	-.509E+06	.000E+00	.129E-01-26275.	0.	-2299.	
.0012	21	332	-.509E+06	.000E+00	.154E-01*****	0.	37079.	
.0013	21	353	-.562E+06	.000E+00	.154E-01*****	0.	-1913.	
.0014	21	394	-.562E+06	.000E+00	.179E-01*****	0.	27721.	
.0015	21	415	-.562E+06	.000E+00	.179E-01*****	0.	-13143.	
.0016	21	436	-.562E+06	.000E+00	.204E-01 -4552.	0.	.637411.	
.0017	21	457	-.562E+06	.000E+00	.204E-01 77480.	0.	-26645.	
.0018	21	498	-.558E+06	.000E+00	.229E-01*****	0.	*****	
.0019	21	519	-.558E+06	.000E+00	.229E-01*****	0.	-44822.	
.0020	21	540	-.558E+06	.000E+00	.254E-01*****	0.	*****	
.0021	21	581	-.558E+06	.000E+00	.254E-01*****	0.	-68433.	
.0022	21	602	-.237E+07	.000E+00	.279E-01*****	0.	*****	
.0023	21	623	-.237E+07	.000E+00	.279E-01	0.	*****	
.0024	21	644	-.237E+07	.000E+00	.304E-01*****	0.	*****	

If you did not look at the screen information during the run, you will likely only check the final displayed results. If you have lots of confidence in the model (and yourself !!) and don't check everything, you may overlook errors.

Suppose that you are only interested in free drainage flow. You check atmospheric flow, but instead of checking the actual atmospheric flow, you only check the potential flow. The latter looks correct as shown (next page); in fact the potential atmospheric flow is always correct since it represents what could potentially go through the boundary, not what actually entered the boundary.



The free drainage flux also looks reasonable if you do not check its magnitude. A quick assessment of the reported value suggests that it must be wrong since it is 5 times the potential atmospheric flux (initial soil profile was dry so it can't be higher than the potential atmospheric flux). Close examination of the actual atmospheric flux shows the source of the error.

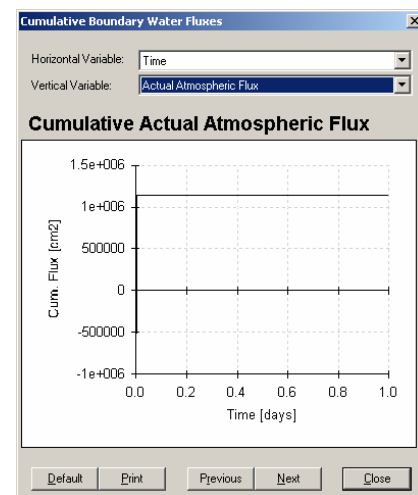
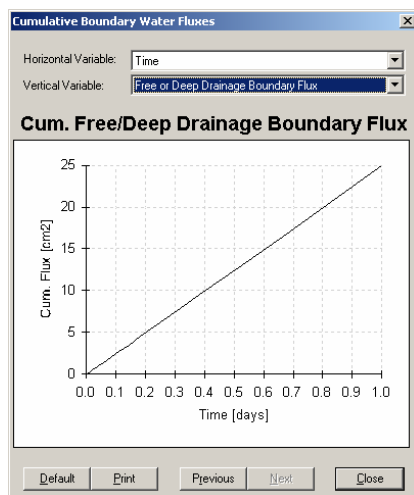


Table 6.1 shows that the mass balance is very poor. Notice that the run time was very short (even shorter than Project 'Grad3'). When the initial conditions are unstable and the initial time step is very small, one should expect a much longer run time. Hence, if the simulation is unexpectedly short, that should be a warning.

▪ Projects 'Grad-5'

In simulation 'Grad5', the initial time step is further reduced to 0.00001 day. Table 6.1 indicates that mass balance results improved, but the errors are still significant. Further reduction in the initial time step does not improve the results. Notice the significant increase in run time (also notice that the simulation time is only 0.1 day,

which is 1/10 of the previous runs). The screen capture at the beginning of the run indicates divergence of the solution during the first two time steps, which explains why the mass balance results were not good.

The screenshot shows a window titled "Hydrus-2D Calculation: grad5" with the following text and data table:

```

reading time information
reading atmospheric information
beginning of numerical solution

```

Time	ItW	ItCum	CumAtmBC	CumConst	CumDrain	hAtm	hConst	hDrain
.100000E-04	21	21	-.500E-04	.000E+00	.250E-03	-1535.	0.	-48.
.200000E-04	21	42	-.100E-03	.000E+00	.499E-03	-1750.	0.	-48.
.300000E-04	8	50	-.150E-03	.000E+00	.502E-03	-1220.	0.	-48.
.400000E-04	9	59	-.200E-03	.000E+00	.505E-03	-1010.	0.	-48.
.500000E-04	7	66	-.250E-03	.000E+00	.508E-03	-875.	0.	-48.
.600000E-04	13	79	-.300E-03	.000E+00	.511E-03	-771.	0.	-48.
.700000E-04	7	86	-.350E-03	.000E+00	.514E-03	-703.	0.	-49.
.800000E-04	7	93	-.400E-03	.000E+00	.517E-03	-646.	0.	-49.
.900000E-04	7	100	-.450E-03	.000E+00	.519E-03	-606.	0.	-50.
.0001	7	107	-.500E-03	.000E+00	.522E-03	-569.	0.	-50.
.0001	11	118	-.550E-03	.000E+00	.525E-03	-540.	0.	-50.
.0001	10	128	-.600E-03	.000E+00	.527E-03	-521.	0.	-51.
.0001	7	135	-.650E-03	.000E+00	.530E-03	-503.	0.	-51.
.0001	7	142	-.700E-03	.000E+00	.532E-03	-485.	0.	-51.
.0002	6	148	-.750E-03	.000E+00	.535E-03	-469.	0.	-52.
.0002	6	154	-.800E-03	.000E+00	.537E-03	-452.	0.	-52.
.0002	8	162	-.850E-03	.000E+00	.540E-03	-440.	0.	-53.
.0002	6	168	-.900E-03	.000E+00	.542E-03	-428.	0.	-53.
.0002	9	177	-.950E-03	.000E+00	.545E-03	-418.	0.	-53.

Projects 'Grad-6'

For simulation 'Grad6', the total number of elements was increased to 201; this resulted in a surface element of only 0.01 cm. Table 6.1 indicates that the mass balance results are now satisfactory. The run time has increased significantly. Figure 6.3 shows that the cumulative free drainage flux is different for the two simulations.

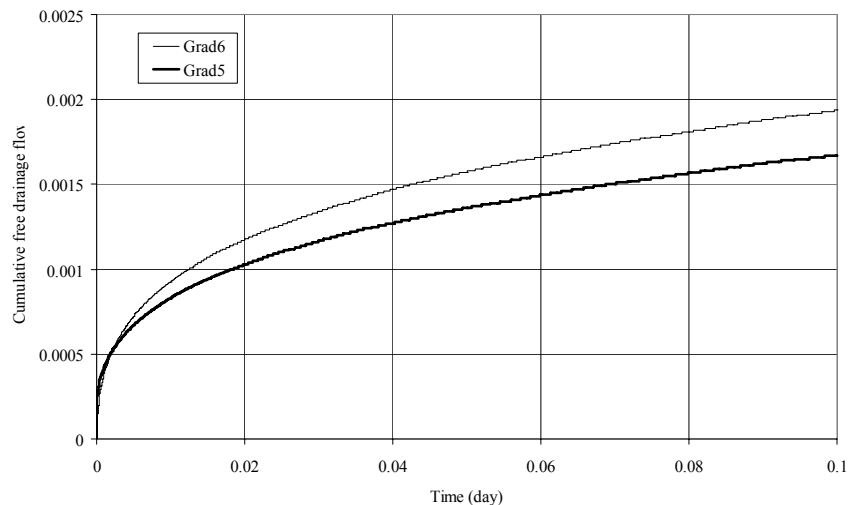


Figure 6.3. Free drainage flow for runs 'Grad5' and 'Grad6'

Hence, one may expect that the presence of very steep pressure gradients will cause more difficulties in getting the model to run smoothly. In addition, the run time should also increase dramatically.

6.6 Concluding Remarks

1. At the conclusion of a simulation, always inspect mass balance and run time information; also make sure that reported fluxes are realistic. Whenever the maximum number of iterations is used, closely inspect the results for possible inconsistencies.
2. For each condition, try to get an optimum combination of mesh size and time stepping scheme that runs as fast as possible, while still producing good mass balance results.
3. Avoid initially fully saturated profiles; they may result in a diverging numerical solution (refer to Section 2.2.1); setting a few surface nodes at a very low negative pressure heads usually helps avoiding the problem.
4. Avoid introducing fluxes that are equal or higher than K_s into the domain.
5. Use the van Genuchten model with $AEV = -2$ cm for fine-textured clayey soils.
6. Some inverse simulations may stall; providing realistic minimum and maximum limits for the optimized parameters should overcome this problem.
7. Complex simulations that run unexpectedly fast indicate a problem.
8. It is always a good idea to monitor the advance of the simulation; screens similar to those shown in pages 6.12 and 6.17 indicate errors. Such simulation may well be terminated using 'Ctrl_C'.
9. Extremely slow simulations usually indicate a flawed set up; check minimum and initial time steps, the density of the grid, and the suitability of the hydraulic model.
10. It is always a good idea to allow small initial and minimum time steps (in the order of seconds).
11. Introducing intense precipitation into an extremely dry soil profile may be problematic (refer to Section 6.2).
12. Leave the default 'Time Step Control' unchanged; in most cases divergence problems are solved by changing the mesh density, the time discretization, and the iteration criteria controls.
13. Using unrealistic initial pressure head profiles is problematic; either use stable initial conditions such as hydrostatic equilibrium or import initial conditions from previous simulations when available.

REFERENCES

- Brooks, R.H. and Corey, A.T. (1966). Properties of porous media affecting fluid flow. *Journal of the Irrigation and Drainage Division ASCE*, 92(IR2): 61-88.
- Brooks, R.H., and Corey, A.T. (1964). Hydraulic properties of porous media. Hydrology paper No. 3, Civil Engineering Department, Colorado State University, Fort Collins, Colorado.
- Campbell, G.S. (1985). *Soil physics with BASIC*, Elsevier, New York.
- Carsel, R. F., and Parrish, R. S. (1988). Developing joint probability distributions of soil water retention characteristics, *Water Resources Research*, 24: 755-769.
- Collis-George, N. (1955). Hysteresis in moisture content-suction relationships in soils. *Proceedings of the National Academy of Science (India)*, 24a: 80-85.
- Croney, D. (1952). The movement and distribution of water in soils. *Geotechnique*, 3(1): 1-16.
- Croney, D., and Coleman, J.D. (1961). Pore pressure and suction in soils. *Proceedings of the Conference on Pore Pressure and Suction in Soils*. Butterworths, London: 31-37.
- Edlefsen NE, and Anderson, A.B.C. (1943). Thermodynamics of Soil Moisture. *Hilgardia*, 15(2): 31-298.
- Everett, EG. (1993). Vadose zone monitoring. In *Geotechnical Practice for Waste Management*, Ed. D.E. Daniel, Chapman and Hall, London: 651-675.
- Feddes, R.A., Kowalik, P.J., and Zarandny, H. (1978). *Simulation of field water use and crop yield*, John Wiley & Sons, New York, NY.
- Gray (1970). *Handbook on the principles of hydrology*. Canadian National Committee for the International Hydrological Decade, National Research Council of Canada, Ottawa.
- Hopmans, J.W. (1987). A comparison of various methods to scale soil hydraulic properties. *Journal of Hydrology*, 93: 241-256.
- Hopmans, J.W., and Stricker, J.N. (1989) Stochastic analysis of soil water regime in a watershed, *Journal of Hydrology*, 105, 57-84.
- Hopmans, J. W., Šimůnek, J., Romano, N., and Durner, W. (2002). 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.
- Koorevaar, P., Menelik, G., and Dirksen, C. (1983). *Elements of soil physics*, Elsevier Science Publications, Amsterdam, The Netherlands.
- Kutilek, M, and Nielsen, D. (1994). *Soil Hydrology*. Cremlingen-Destedt, Germany.
- Miller, E.E., and Miller, R.D. (1956). Physical theory for capillary flow phenomena, *Journal of Applied Physics*, 27: 324-332.

- Mualem, Y. (1976). A new model for predicting the hydraulic conductivity of unsaturated porous media. *Water Resources Engineering*, 12: 513-522.
- Peck, A.J., Luxmoore, R.J., and Stolzy, J.L. (1977). Effects of spatial variability of soil hydraulic properties in water budget modelling. *Water Resources Research*, 13(2): 348-354.
- Rancic, A. (2002). Personal communications. Department of Land and Water Conservation, NSW, Australia.
- Rassam, D.W., and Williams, D.J. (1999a). Unsaturated hydraulic conductivity of mine tailings under wetting and drying conditions. *Geotechnical Testing Journal ASTM*, 22: 138-146.
- Rassam, D.W., and Williams, D.J. (1999b). A numerical study of steady-state evaporative conditions applied to mine tailings. *Canadian Geotechnical Journal*, 36: 640-650.
- Rawls, W. J., Brakensiek, D. L., and Saxton, K. E., 1982, Estimating soil water properties, *Transactions, ASAE*, 25(5), 1316-1320 and 1328.
- Richards, B.G. (1965). Determination of the unsaturated permeability and diffusivity functions from pressure plate outflow data with non-negligible membrane impedance. In G.D. Aitchison (Ed.), *Moisture equilibria and moisture changes in soils beneath covered areas*, Butterworths, Australia: 47-54.
- Šimůnek, J., Kodesova, R., Gribb, M., and van Genuchten, M. Th. (1999). Estimating hysteresis in the soil water retention function from cone permeameter experiments. *Water Resources Research*, 35(5): 1329-1345.
- Šimůnek, J., Sejna, M., and van Genuchten, M. Th. (1999). The HYDRUS-2D software package for simulating two-dimensional movement of water, heat, and multiple solutes in variably saturated media. Version 2.0, IGWMC - TPS - 53, International Ground Water Modeling Center, Colorado School of Mines, Golden, Colorado.
- Šimůnek, J., and Hopmans, J. W. (2002). 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.
- Šimůnek, J., Jacques, D., Hopmans, J. W., Inoue, M., Flury, M., and van Genuchten, M. Th. (2002). 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.
- Taylor, S. A., and Ashcroft, G. M. (1972). *Physical Edaphology*. Freeman and Co., San Francisco, California, p. 434-435.
- Tinjum, J.M., Benson, C.H., and Blotz, L.R (1997). "Soil-water characteristic curves for compacted clays." *Journal of Geotechnical and Geoenvironmental Engineering, ASCE*, 123(11): 1060-1069.
- van Genuchten, M. Th. (1987). A numerical model for water and solute movement in and below the root zone. Research Report N. 121, U.S. Salinity Laboratory, USDA, ARS, Riverside, California.

- van Genuchten, M.Th. (1980). A closed-form equation for predicting the hydraulic conductivity of unsaturated soils. *Soil Science Society of America Journal*, 44: 892-898.
- Vogel, T., and Cislserova, M. (1988). On the reliability of unsaturated hydraulic conductivity calculated from the moisture retention curve, *Transport in Porous Media*, 3: 1-15.
- Vogel, T., Cislserova, M., and Hopmans, J.W. (1991). Porous media with linearly variable hydraulic properties. *Water Resources Research*, 27(10): 2735-2741.
- Vrugt, J.A., Hopmans, J.W., and Simunek, J. (2001). Calibration of a two-dimensional root water uptake model. *Soil Science Society of America Journal*, 65: 1027-1037.
- Wesseling, J. G. (1991). Meerjarige simulaties van grondwateronttrekking voor verschillende bodemprofielen, grondwatertrappen en gewassen met het model SWTRE. Report 152, Winand Staring Centre, Wageningen.
- Wilson, G.W., Fredlund, D.G., and Barbour, S.L. (1997). The effect of soil suction on evaporative fluxes from soil surfaces. *Canadian Geotechnical Journal*, 34: 145-155.

APPENDIX I

SOIL HYDRAULIC PROPERTIES

I.1 Water Retention Properties

The fundamental relationship between soil moisture content (usually calculated on a volumetric basis), and pressure head (or matric suction) is referred to as the soil-water retention curve (SWRC), or the soil water-characteristic curve. Matric suction is the difference between pore air pressure and pore water pressure. If a change in moisture content occurs at any point in a mass of soil initially in a state of moisture equilibrium, the soil matric suction (or the negative pore water pressure) at that point changes (assuming atmospheric conditions, pore air pressure = 0). Following this change, there will be a differential in the energy level between the point where the change occurred and the surrounding soil, which will cause moisture to flow in the soil until a new condition of suction equilibrium is established (Croney, 1952). Knowledge of the water retention curve is essential when investigating how soil moisture reacts following changes in the pressure head resulting from events such as infiltration, evaporation and drainage. Due to the hysteretic behaviour of soils upon wetting and drying, different SWRC curves are obtained depending upon whether the soil is undergoing a drying cycle, a wetting cycle, or a re-drying cycle (Collis-George, 1955).

I.1.1 Shape of Soil-Water Retention Curve

The general shape of the SWRC under wetting and drying conditions is shown in Figure I.1 (semi-log scale). The following parameters are identified:

- The saturated volumetric water content θ_s , which theoretically corresponds to the soil's porosity, but practically is often about 10-25% less because of dissolved and/or entrapped air.
- The air-entry value, AEV, is the critical suction value at which the largest pores in the soil matrix begin to lose water.
- The residual water content θ_r , is the water content beyond which further increase in the soil's suction results in only marginal changes in water content.

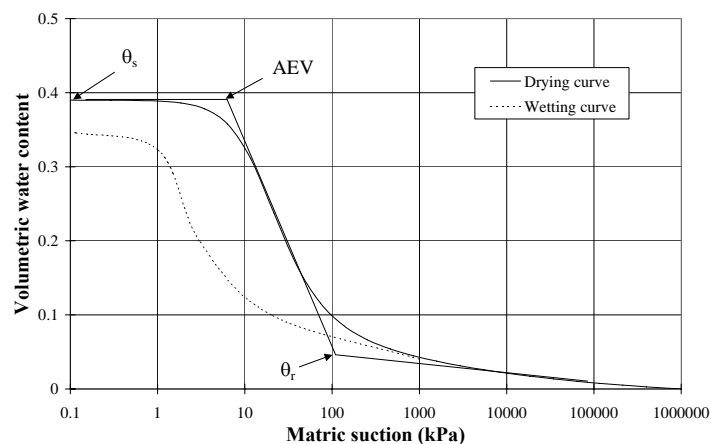


Figure I.1: Typical SWRC

The total suction corresponding to zero water content is essentially the same for all soils. This value is about 10^6 kPa (10^7 cm) and has been supported experimentally by Cronney and Coleman (1961), Koorevaar et al. (1983), and by thermodynamic considerations (Richards, 1965).

Definition of the Residual Water Content

The residual water content and the corresponding suction, termed residual suction, have been defined in a number of ways in the literature; however, the accepted use of these terms is not always agreed upon. Tinjum et al. (1997) defined it as the water content corresponding to the asymptote of the SWRC at low degrees of saturation. van Genuchten (1980) defined it as the water content for which its rate of change with respect to suction becomes zero, or when liquid flow of water ceases (excluding the region near saturation); he suggested that often it is sufficient to define it as the water content at a suction equal to 1500 kPa.

Classically, moisture in the unsaturated zone is separated into three components: gravity, pellicular, and hygroscopic water (Everett 1993). Gravity water is the moisture in a soil that can be drained by gravitational forces; pellicular water is the moisture in a soil that cannot be drained by gravity forces, but can be lost by evaporation; and hygroscopic water is moisture that will never be lost through the above natural forces. The residual water content is the water content at the limit of liquid water extraction. De-saturation beyond residual conditions occurs primarily as a result of vapor movement up to the point where the soil water content is in equilibrium with the vapor pressure of its surrounding. Brooks and Corey (1966) pointed out that it is difficult to determine residual saturation for clayey materials; they related residual saturation to the clay content of the soil. The well-defined, residual state for sands, silts, and their mixtures can reasonably well be predicted from the SWRC as shown in Figure I.1.

1.1.2 Models Describing SWRC

A mathematical representation of the SWRC is essential when using numerical models for simulating fluid flow and mass transfer in the unsaturated zone. HYDRUS allows users to select one of three types of models to describe the soil hydraulic properties: van Genuchten (1980), modified van Genuchten (Vogel and Cislerova, 1988), and Brooks and Corey (1964). Those three models are described briefly in the following section.

a. van Genuchten Model

Van Genuchten (1980) proposed a mathematical representation of the SWRC, which depicts an S-shaped curve when plotted on a semi-log scale. It is given by:

$$S_e = \left(1 + |\alpha\psi|^n\right)^{-m} \quad (I.1)$$

where α , m , and n are fitting parameters (usually $m=1-1/n$), and S_e is the normalised volumetric water content (also called effective saturation) given by:

$$S_e = \frac{\theta - \theta_r}{\theta_s - \theta_r} \quad (\text{I.2})$$

where θ is the volumetric water content at any pressure head, and θ_s and θ_r are the saturated and residual water contents, respectively.

Each of the three fitting parameters (α , m , and n) control part of the S-shape curve. Figure I.2 shows the effect of α (Alpha), which is closely related to the reciprocal of the air-entry value (AEV) of the soil. Coarse-grained soils have a low AEV and high α whereas fine-textured soils have a lower α .

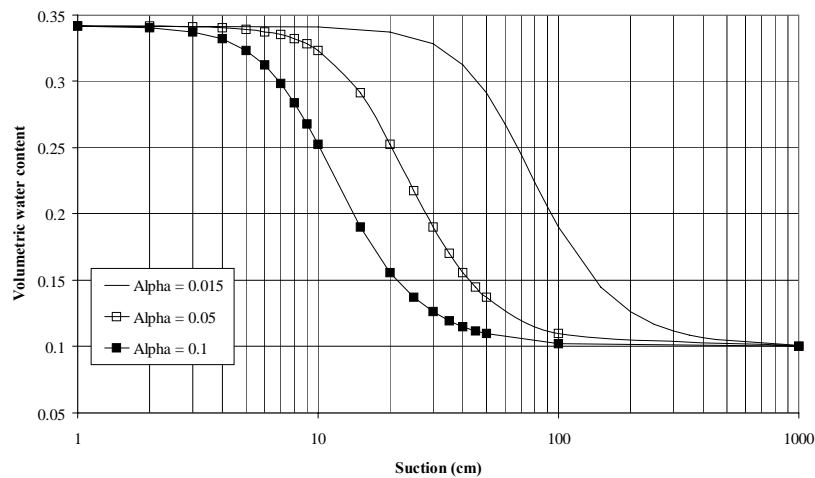


Figure I.2: Effect of α in van Genuchten model

Figure I.3 shows the effect of n , which controls the slope of the curve and reflects the particle size distribution. The plots indicate that a unit increase in suction at or near the steepest part of the curve causes more water to be extracted from coarse-textured soils (high n) than from fine-textured soils (low n), but not at other places of the curve.

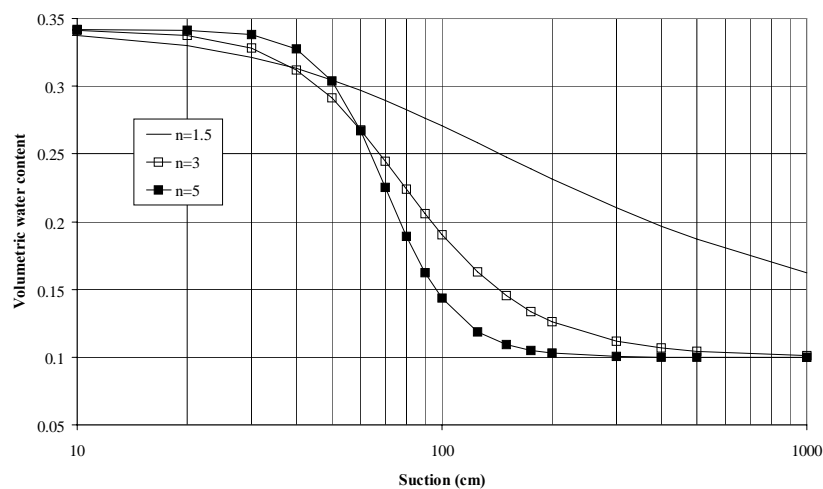


Figure I.3: Effect of n in van Genuchten model

The parameters n and m are usually correlated as $m=1-1/n$, which results in a relatively simple equation for the hydraulic conductivity when combined with a theoretical model for K (see section I.2). Figure I.4 shows that when the two parameters are correlated, the curvatures (A and B) of the S-shape curve become correlated. When individual values are assigned to n and m the curvatures become independent.

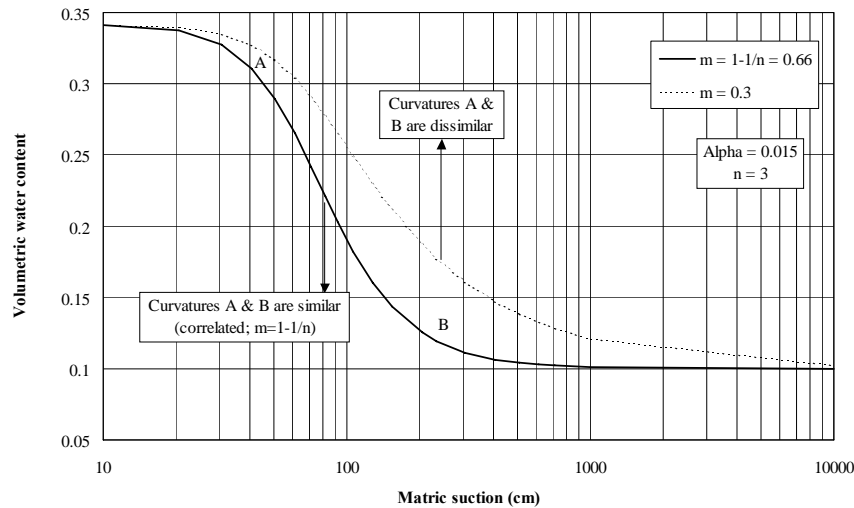


Figure I.4: Relation of m and n in van Genuchten model

b. van Genuchten Model with AEV of -2 cm

This option in the van Genuchten model implements a small non-zero air-entry value (AEV) of -2 cm. Figure I.5 shows that the effect is very marginal on the SWRC. In fact one can only notice the difference when that part near saturation is significantly enlarged. Figure I.5 shows that the water content remains at saturation for suctions below 2 cm.

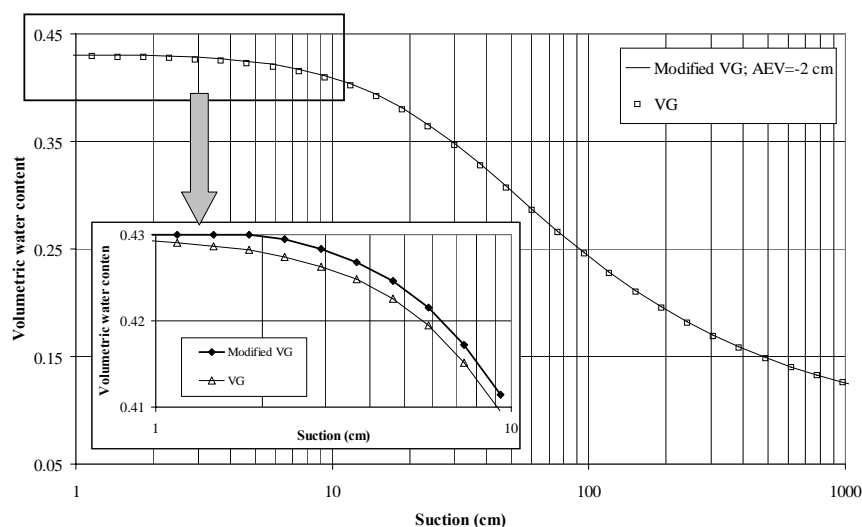


Figure I.5: van Genuchten model with AEV of -2 cm

We will show later that implementing this low non-zero AEV has a significant effect on the hydraulic conductivity function. This option is particularly relevant to very fine-textured (clayey) soils (see example in section 6.4).

c. Modified van Genuchten Model (Vogel and Cislérova)

Vogel and Cislérova (1988) modified the van Genuchten model by incorporating a non-zero AEV into the model. The modification is implemented by introducing a fictitious water content θ_m that is higher than θ_s and replaces θ_s in the van Genuchten model. This fictitious water content is used only when $h < \text{AEV}$. Above AEV the water content is equal to θ_s .

$$\theta = \begin{cases} \theta_a + \frac{\theta_m - \theta_a}{\left(1 + |\alpha h^n|\right)^{-m}} & h < \text{AEV} \\ \theta_s & h \geq \text{AEV} \end{cases} \quad (\text{I.3})$$

Figure I.6 shows how the model is implemented. The data points at suctions < 17 cm are only a hypothetical continuation to the S-shape curve. That is, when the head is below this value, the water content is fixed at θ_s .

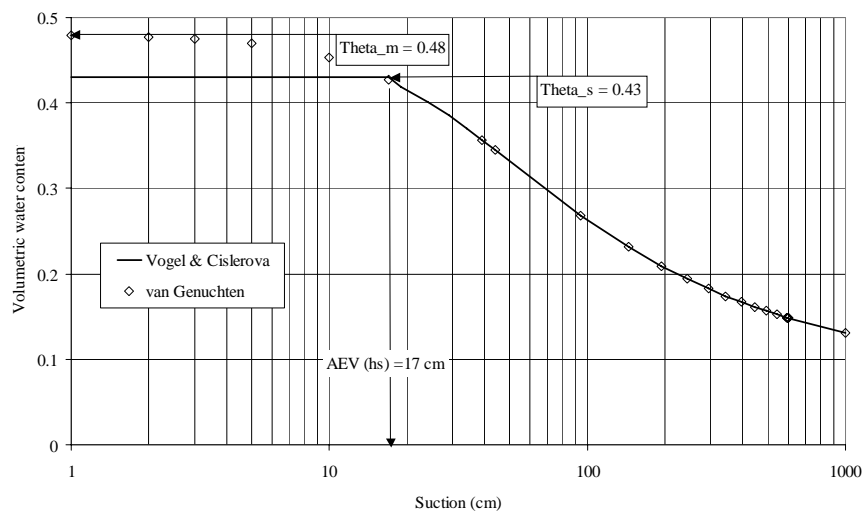


Figure I.6: Modified van Genuchten model

d. Brooks and Corey Model

Brooks and Corey (1964, 1966) concluded from comparisons of a large number of experimental data that the SWRC could be described by the following formula:

$$S_e = \left(\frac{\text{AEV}}{h} \right)^n \quad (\text{I.4})$$

where AEV is the air entry value (corresponds approximately to $1/\alpha$ in HYDRUS), and n is a soil characteristic parameter, called the pore-size-distribution index. Experimental data of suction and water content are often plotted as $\log |h|$ versus \log

θ . Linear regression can then be performed on the straight-line relationship. The slope and intercept of the best-fit line correspond to $1/n$ and AEV, respectively. Figure I.7 shows a comparison of this model with the van Genuchten model using the parameters listed in Table I.1.

Table I.1: Parameters for two SWRC for Figure I.7.

	Theta_r	Theta_s	Alpha	n
Van Genuchten	0.078	0.43	0.036	1.56
Brooks & Corey	0.027	0.434	0.0897	0.293

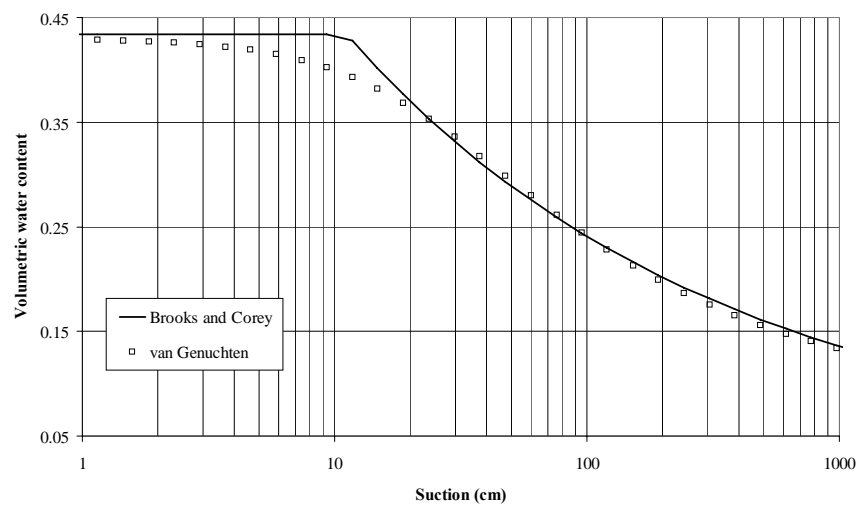


Figure I.7: Comparison of Brooks & Corey and van Genuchten models

I.2 Hydraulic Conductivity Functions

a. van Genuchten-Mualem Model

Mualem (1976) developed the following equation for predicting the relative hydraulic conductivity K_r from knowledge of the SWRC:

$$k_r = \sqrt{S_e} \left(\frac{\int_0^{\theta} \frac{1}{h(x)} dx}{\int_0^1 \frac{1}{h(x)} dx} \right)^2 \quad (\text{I.5})$$

where h is the pressure head, given as a function of the dimensionless water content S_e defined by Equation (I.2).

For the special case where $m = 1 - 1/n$ (fitting parameters of Equation (I.1)), a closed-form solution was obtained by van Genuchten (1980) as follows:

$$k(h) = K_s S_e \left[1 - \left(1 - S_e^{1/m} \right)^m \right]^2 \quad (\text{I.6})$$

where l is a pore connectivity parameter established to be 0.5 by Mualem (1976). Alternatively, this equation may be written in terms of the effective water content (see online HYDRUS Help). When the -2 cm AEV option is used, Equation I.6 becomes applicable only for heads < -2 cm. Figure I.8 shows the impact of using this option. Note that hydraulic conductivity functions are usually plotted to log-log scale, but in this case we chose to plot it on a semi-log scale to emphasize the dramatic differences near saturation.

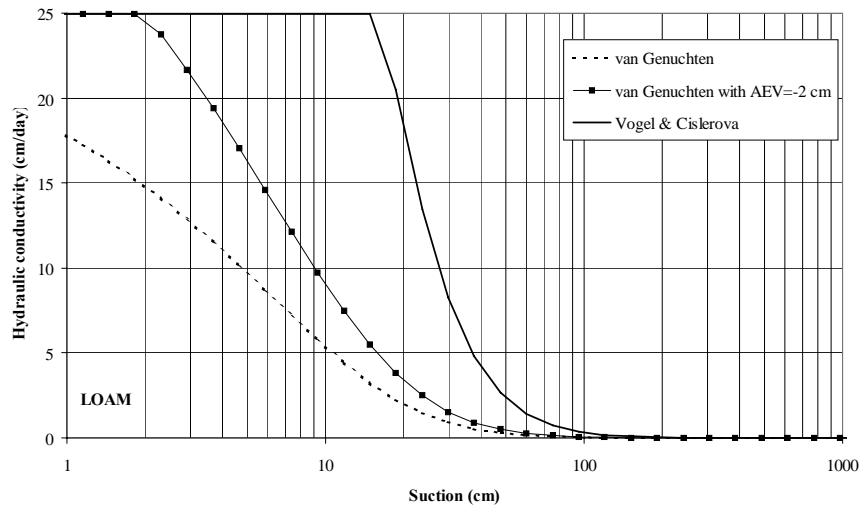


Figure I.8: Effect of non-zero AEV on hydraulic conductivity function

b. Modified van Genuchten Model (Vogel and Cislserova)

When using this model, the hydraulic conductivity function is discretized into three zones as shown in Figure I.9:

- $h \geq h_s$ (AEV) $K(h) = K_s$
- $h_k < h < h_s$ $K(h) = \text{linear value between } K_s \text{ and } K_k$
- $h \leq h_k$ $K(h) = \text{non-linear value (Mualem model)}$

where h_k is the head corresponding to a user-defined water content θ_k (Q_k in HYDRUS); θ_k is water content below saturation where a measurement of the hydraulic conductivity K_k is available. The mathematical formulation for this model is found in the online Help and will not be repeated here. A comparison between the results of this model and the original van Genuchten model is shown in Figure I.10. Figure I.8 shows the significant differences near saturation. The modification was made to enable relatively large increases in K near saturation that are often observed in structured (macroporous) soils.

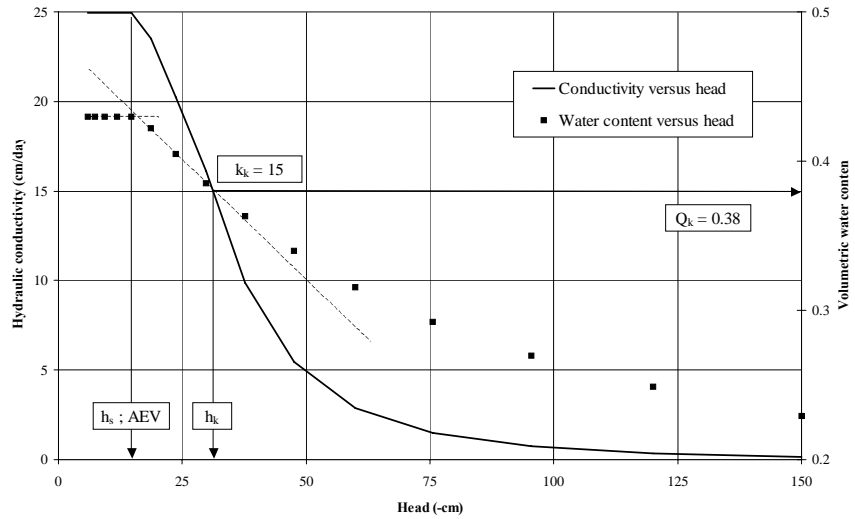


Figure I.9: Vogel and Cislerova hydraulic conductivity function

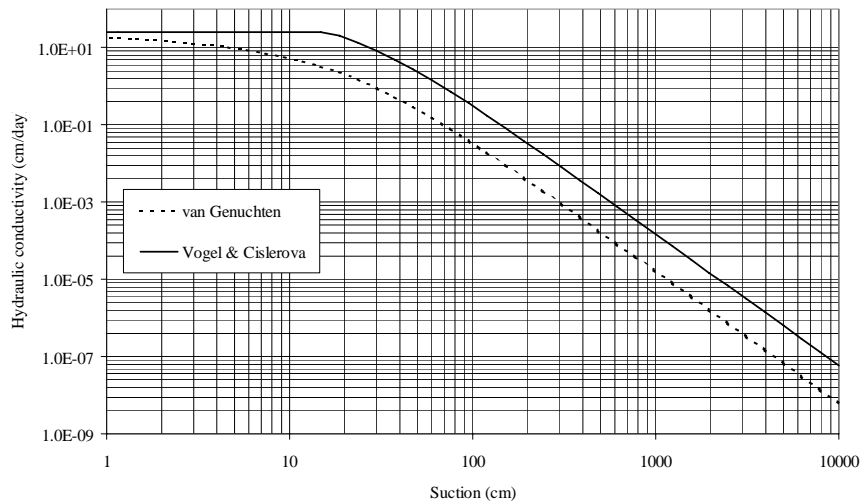


Figure I.10: van Genuchten and Vogel & Cislerova hydraulic conductivity functions

c. Brooks and Corey Model

For heads <AEV, the hydraulic conductivity is defined as follows:

$$k(h) = k_s \left(\frac{AEV}{h} \right)^{2+3n} \tag{I.7}$$

A comparison of the results obtained with this model, and those obtained with the van Genuchten model is shown in Figure I.11. The hydraulic parameters listed in Table I.1 were used for this purpose.

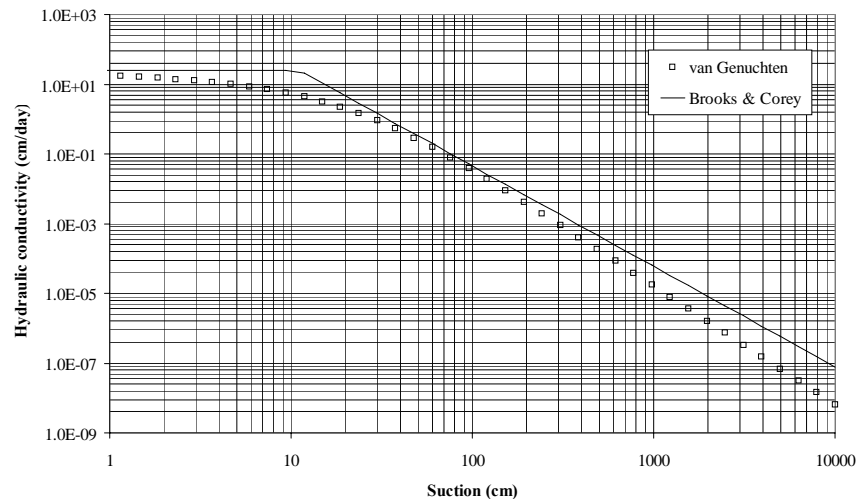


Figure I.11: van Genuchten and Brooks & Corey hydraulic conductivity functions

I.3 Sensitivity Analysis; Understanding How the Hydraulic Parameters of a Soil Can Affect Simulation Results

It is crucial for users to understand how each of the many input parameters impacts the simulation results. If the model is more sensitive to a particular parameter, one should invest more resources and effort into getting a high-confidence estimate for it. This is also crucial when conducting inverse simulations where the task is estimating a set of parameters for the model. Parameters that the model is insensitive to may be omitted from the optimisation list, thus reducing correlations between the other optimised parameters. In addition, modellers may not have good prior knowledge of the soil hydraulic parameters but they still need to come up with realistic initial starting parameters before running the inverse mode.

In this section we will investigate the effect of each of the five hydraulic parameters Alpha, n , K_s , Q_s , and Q_r . This sensitivity analysis is conducted using Project “a3”, section 5.2.1.

This exercise enhances our physical understanding of the effect of various hydraulic parameters. In addition, it helps when we manually choose initial parameters for inverse runs, which may have a great impact on the optimisation result (Section 5.3). That is, if we know the effects of each of the parameters, we can manually change them and run trial forward runs until a reasonable agreement is obtained with the observed data, and then let the inverse solution do the final refinement (see Section 5.3). This is advantageous in two aspects: firstly, it excludes unrealistic results that arise when starting with physically unrealistic initial parameters, and, secondly, it reduces the run time (very advantageous for relatively long, complex simulations).

I.3.1 Parameter Alpha. Increasing this parameter leads to (see Figure I.12):

- Lower initial water contents, since in this case the soils initial condition is represented by the pressure head.

- Later arrival of the wetting boundary.
- Higher final water contents.

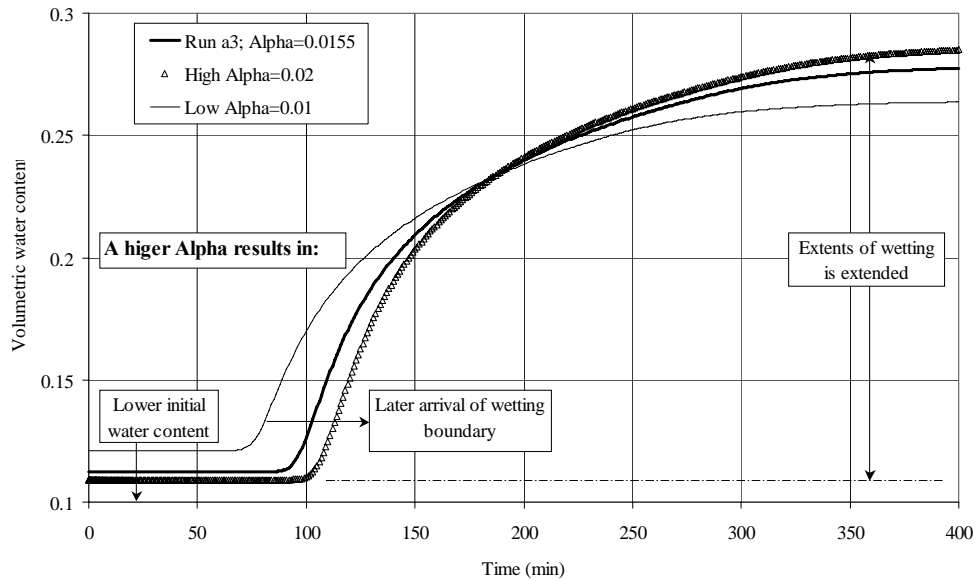


Figure I.12: Effect of parameter Alpha

I.3.2 Parameter n. Increasing this parameter leads to (see Figure I.13):

- Downward translation of the entire curve.

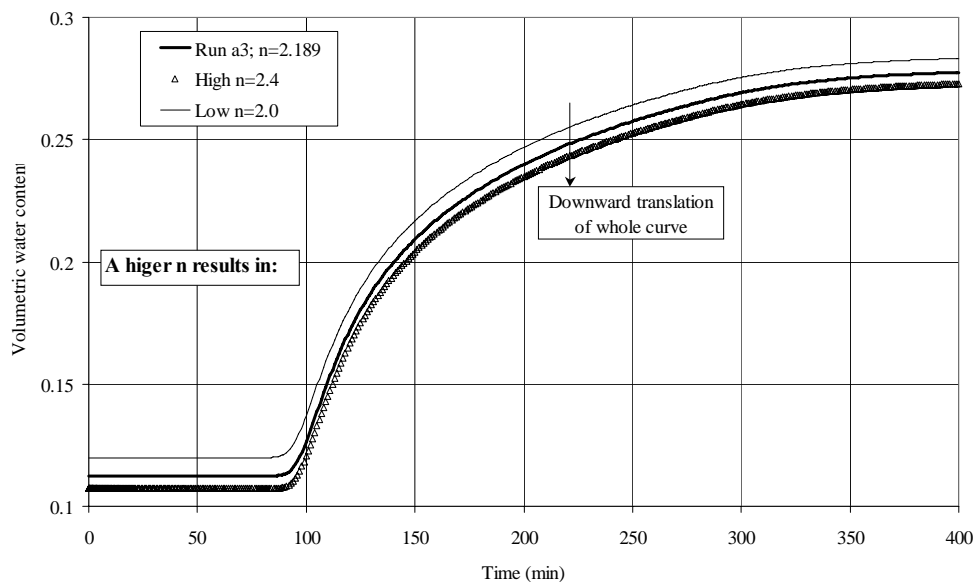
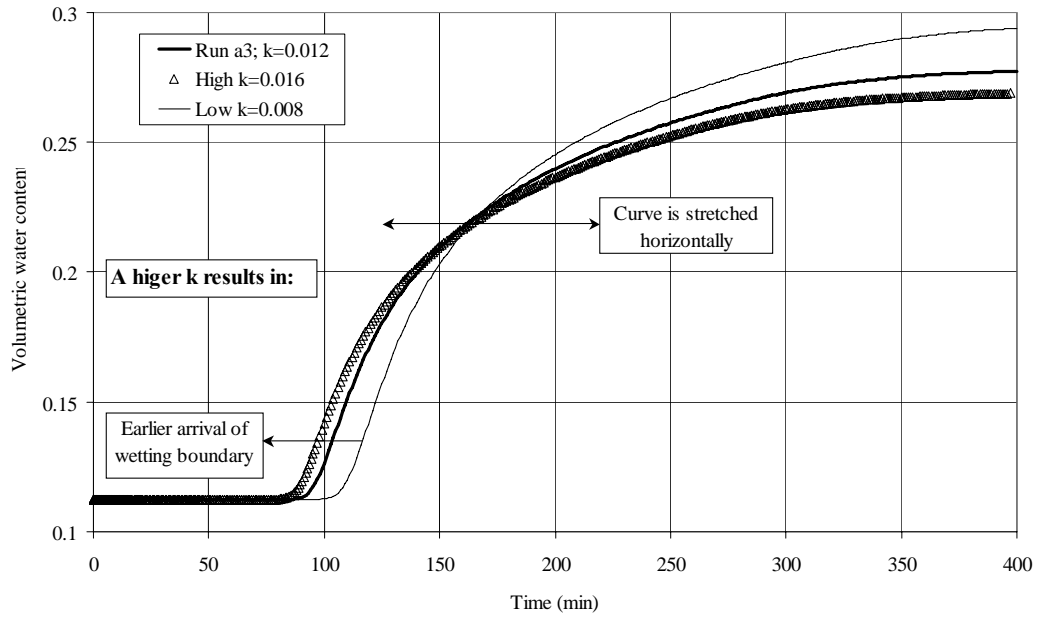


Figure I.13: Effect of parameter n

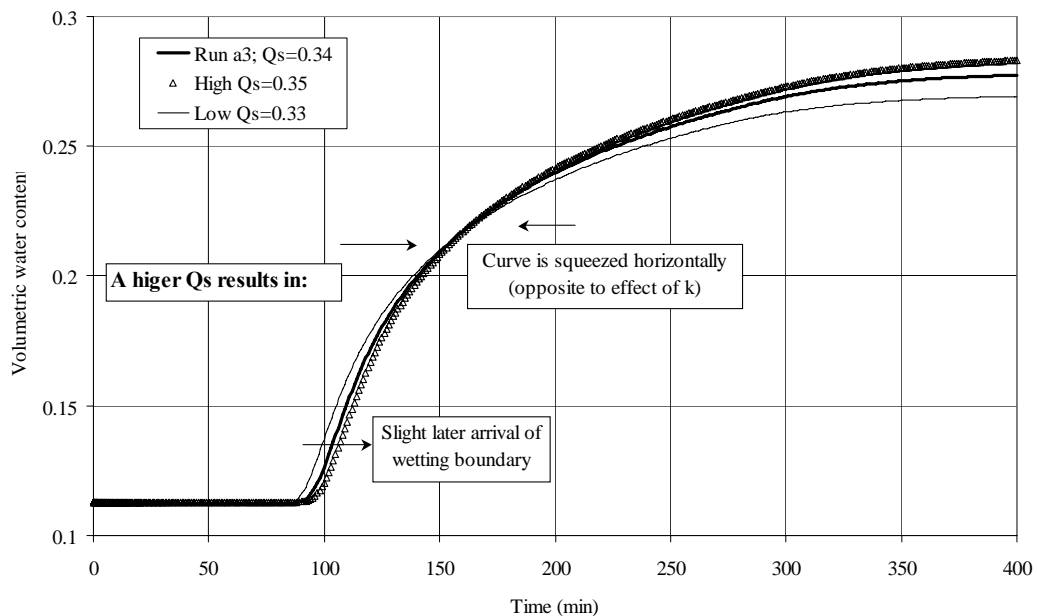
I.3.3 Parameter Ks. Increasing this parameter leads to (see Figure I.14):

- Earlier arrival of the wetting boundary.
- Lower final water contents.

Figure I.14: Effect of parameter k

I.3.4 Parameter θ_s . Increasing this parameter leads to (see Figure I.15):

- Slightly later arrival of wetting front
- Changes in the shape of the curve in a manner opposite to that when K_s is increased.

Figure I.15: Effect of parameter Q_s

I.3.5 Parameter θ_r . Increasing this parameter leads to (see Figure I.16):

- Higher initial water contents.
- Earlier arrival of the wetting front.

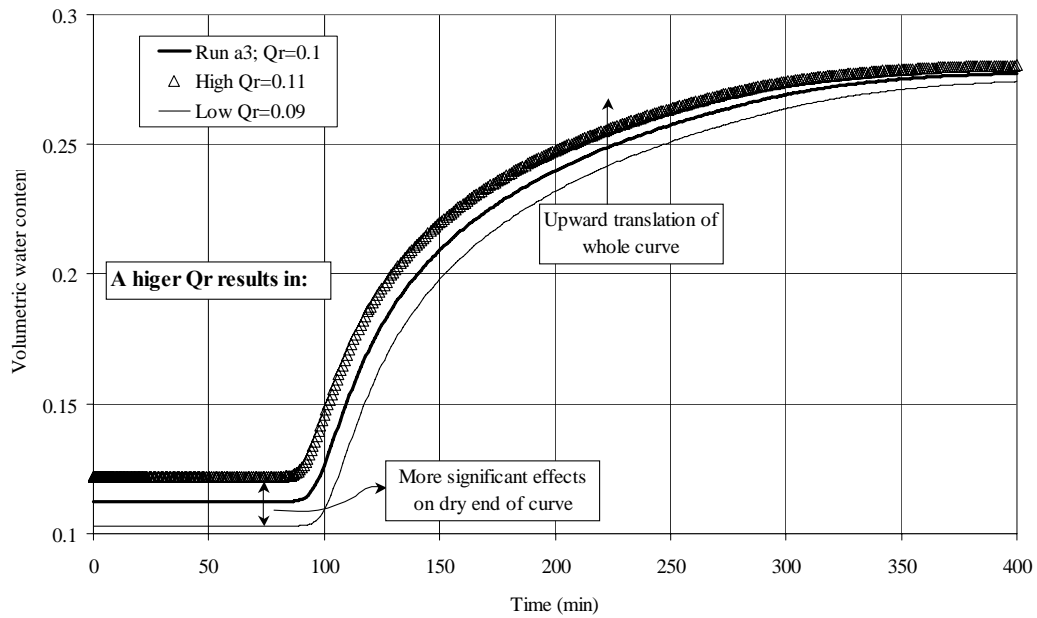


Figure I.16: Effect of parameter Q_r

APPENDIX II

CONCEPTS RELATED TO MODELLING EVAPORATION

Evaporation from a soil surface is an important component of the water budget. Evaporation rates are influenced by external weather conditions as well as soil parameters. Weather conditions usually dictate what is known as “potential evaporation”, which is the maximum rate of evaporation from a pure water surface. Figure II.1 shows the shape of the drying curve during evaporation, which comprises three stages. During the first stage, the evaporation rate is relatively constant, and the soil can fully deliver water to keep the potential rate, yet remains at or near saturation (actual evaporation = potential evaporation). The second stage commences when the soil surface starts to de-saturate and can no longer supply water at the rate dictated by atmospheric demand (actual evaporation < potential evaporation). At this stage, the soil hydraulic and water retention parameters start to play an important role in the evaporation process. As drying proceeds, the evaporation rate continues to decline as more water is extracted from the soil and the contribution of the liquid-water phase to flow starts to diminish. During the third stage of evaporation, the water phase becomes discontinuous and water flow occurs predominantly through vapour diffusion.

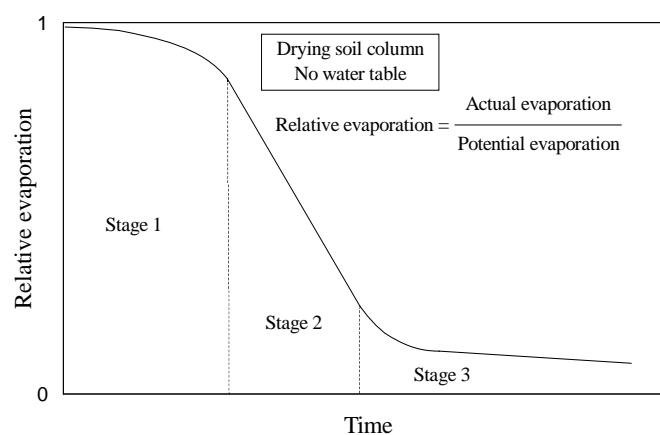


Figure II.1: Drying curve during evaporation

II.1 Significance of Suctions at the Soil Surface

HYDRUS does not model vapour flow, but uses a mathematical scheme that mimics the stages of evaporation. In this section we briefly review the significance of soil suctions at the evaporating soil surface, and how they relate to vapour flow and relative humidity.

Based on a mass transfer equation proposed by Dalton in 1802 (reported by Gray, (1970)), actual evaporation E_a from a soil surface may be estimated using the following formula (Campbell, 1985), which estimates the actual evaporative flux from a soil surface in response to a gradient in the relative humidity between the soil and the atmosphere:

$$E_a = \frac{H_s - H_a}{1 - H_a} E_p \quad (\text{II.1})$$

where E_p is the potential or pan evaporation rate, H_s is the humidity at the soil surface calculated using Eq. (II.2), and H_a is the atmospheric humidity. Relative humidity is related to soil's total suction as follows (Edlefsen and Anderson, 1943):

$$H_r = \exp\left(\frac{M_w \psi}{\Theta R \rho_w}\right) \quad (\text{II.2})$$

where M_w is the mass of a mole of water (0.018 kg/mol), ψ is the soil-water potential in Pa (Pa is Newton/m²; Newton=J/m; 1 kPa=1000 pa= 10 cm water pressure head), R is the gas constant (8.3143 J/mole K), Θ is the absolute temperature (K; = °C+273), and ρ_w is the density of water (998 kg/m³ at 20°C). Assuming a mean temperature of 20 °C, a total suction of 300 m corresponds to a 97.8% relative humidity. Equation II.2 is plotted in Figure II.2.

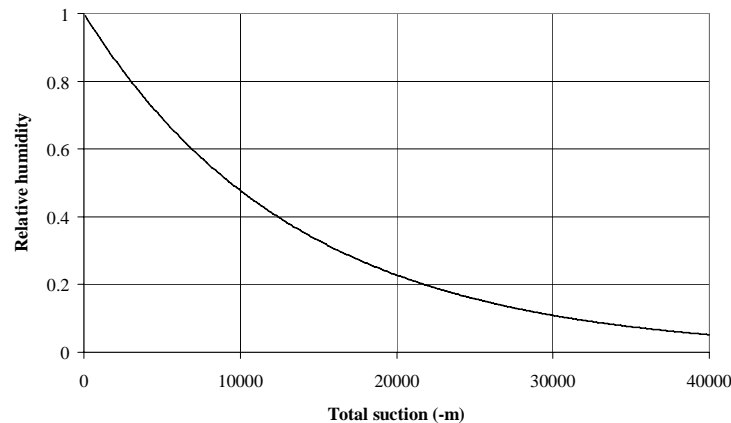


Figure II.2: Relation of total suction to relative humidity

Figure II.3 shows a numerical simulation of evaporation with vapour flow. During the 1st stage of evaporation, flow is mainly in the liquid phase. Stage 2 starts when the surface dries and can no longer supply moisture at the potential rate (Time>32 hrs); the vapour component then rapidly increases. Notice how the suctions significantly increase during this stage (Figure II.4; Time>32 hrs). Rassam and Williams (1999b) have shown that the maximum rate of change of the surface relative humidity occurs at a Suction of 300 m (3,000 kPa).

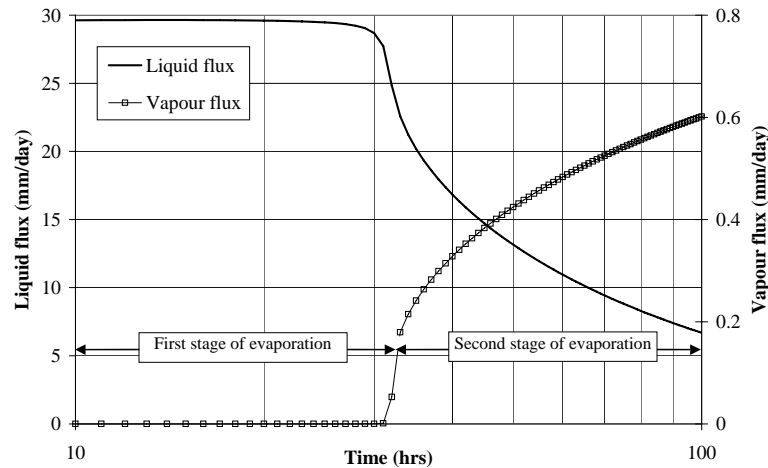


Figure II.3: Liquid- and vapour-phase fluxes during evaporation

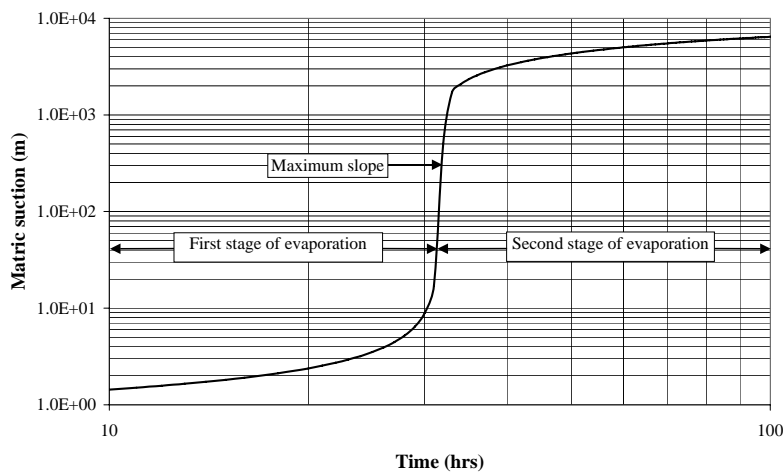


Figure II.4: Development of surface suctions during evaporation

Wilson et al. (1997) experimentally showed that regardless of soil texture, the actual evaporation rate from a soil surface starts to drop below the potential rate when the surface suction reaches 300 m (3,000 kPa, a relative humidity of about 98%).

II.2 Concept of h_{CritA}

HYDRUS implements a scheme whereby the actual evaporative flux remains equal to the potential flux up to some user-defined suction called “ h_{CritA} ”. This value is specified in the “Variable Boundary Condition” as a positive value that represents suction (negative head) using the unit that was adopted in the simulation (m, cm, or mm). This value may vary with soil type and could have a significant effect on the simulation results.

Project “Evap” simulates evaporation from a 3-m deep soil profile with a constant head boundary at the bottom. This project is identical to projects ‘Evap1’ and ‘Evap2’ (CD..HYDRUS Manual Projects(Sect-4)). Various combinations of soil types and h_{CritA} values were used; results are shown in figures II.5, II.6, and II.7. The time at which the evaporative flux falls below potential is hereby termed T_{crit} . In this analysis

we assume that the suitable h_{critA} is that value beyond which no significant change in T_{crit} occurs.

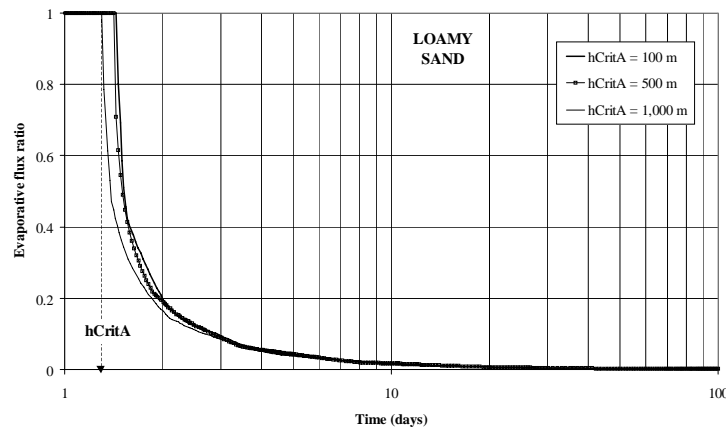


Figure II.5: Effect of h_{critA} on evaporative flux for loamy sand

Figure II.5 shows that the effect of h_{critA} is marginal for a loamy sand (a fairly coarse-texture soil). The trends for values of 100 m and 500 m are almost identical. Figure II.6 shows that for a silty soil, a higher value is warranted. Figure II.7 indicates that a clayey (fine-textured) soil is the most sensitive to h_{critA} . Table II.1 shows the recommended values for several soil classes. Notice the effect of soil type on T_{crit} , where silt continues to provide flux at potential for periods much longer than the two other soil types.

Table II.1: h_{critA} for various soil classes

Soil class	h_{critA} (m)
Sandy	500
Silty	1,000
Clayey	3,000

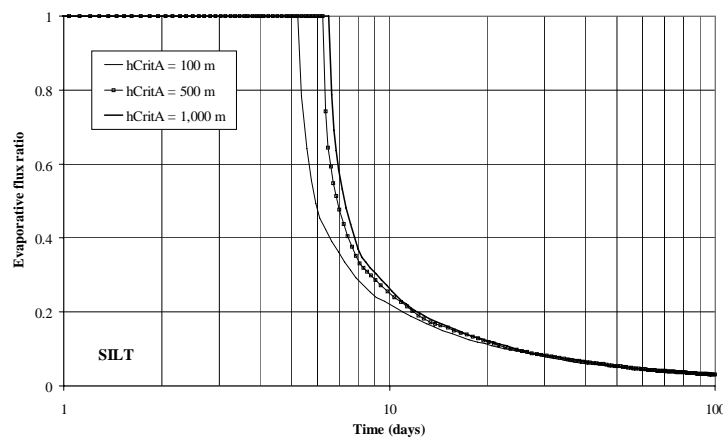


Figure II.6: Effect of h_{critA} on the evaporative flux for a typical silt

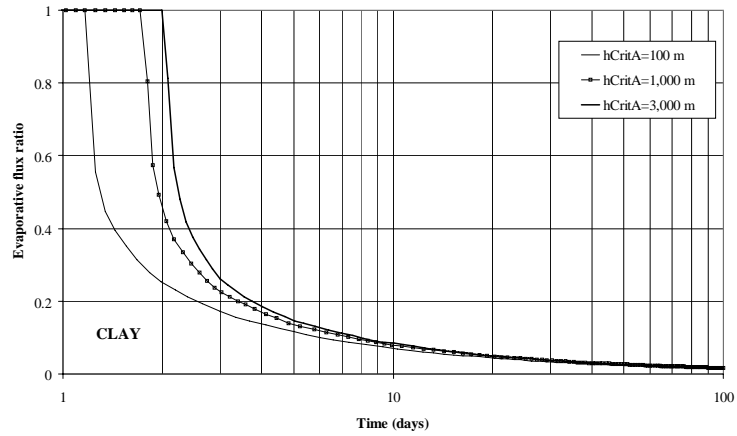


Figure II.7: Effect of hCritA on the evaporative flux for a typical clay

APPENDIX III

ROOT WATER UPTAKE

III.1 Root Water Uptake Reduction Models

The water uptake reduction models define the manner in which transpiration is reduced below potential when the soil is no longer capable of supplying the amount of water required by the plant under the prevailing climatic conditions. There are two alternative models in HYDRUS: the Feddes model (Feddes et al., 1978) and an S-shaped model (van Genuchten, 1987). The former is more commonly used.

Feddes Model Parameters

The Feddes model at any point in the root zone assigns plant root water uptake rates according to the local soil water pressure head. The meaning of the Feddes' parameters is demonstrated in Figure III.1.

P_0 : Value of the pressure head below which roots start to extract water from the soil.

P_{opt} : Value of the pressure head below which roots extract water at the maximum possible rate (potential transpiration).

P_{2H} : Value of the limiting pressure head below which roots no longer extract water at the maximum rate (assuming a potential transpiration rate of r_{2H}).

P_{2L} : As above, but for a potential transpiration rate of r_{2L} .

P_3 : Value of the pressure head below which root water uptake ceases (usually taken at the wilting point).

A database of suggested values for different plants is provided based on studies by Wesseling (1991) and Taylor and Ashcroft (1972).

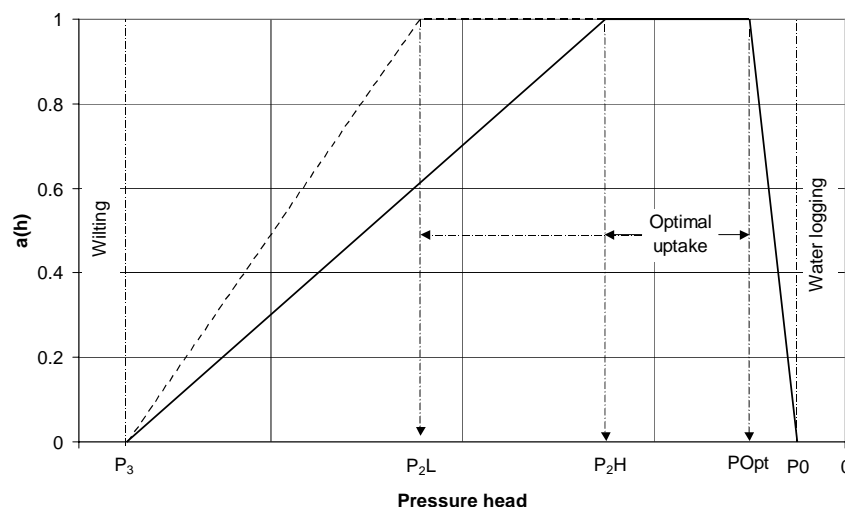


Figure III.1: The Feddes root water uptake reduction model

r2H: Potential transpiration rate (L/T) (currently set at 0.5 cm/day).

r2L: Potential transpiration rate (L/T) (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 (P2 presumably decreases at higher transpiration rates). HYDRUS currently implements a linear interpolation scheme (defined in the HYDRUS Technical Manual).

S-Shaped Model Parameters

van Genuchten (1987) proposed an S-shaped function to describe root water uptake reduction, which is applicable to water stress as well as salinity stress:

$$a(h) = \frac{1}{1 + \left(\frac{P}{P50}\right)^{P3}} \quad (\text{III.1})$$

where P3 is an experimental constant and P50 is the pressure head at which local uptake rate is half the potential rate. Figure III.2 below shows the S-shaped function for a case where P50=-800 cm and P3=3.

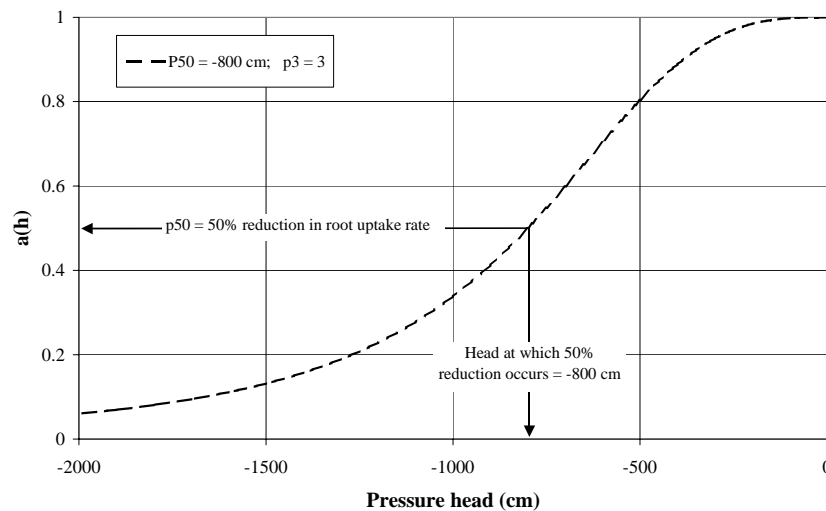


Figure III.2: S-shaped root water uptake reduction model

III.2 Root Density

Relative root densities are assigned to each node in the root zone. Values are either manually assigned to individual nodes or HYDRUS assigns it using a root distribution function. Figure III.3 shows an example of a root distribution along 12 equally spaced nodes representing a 1.2-m deep root zone.

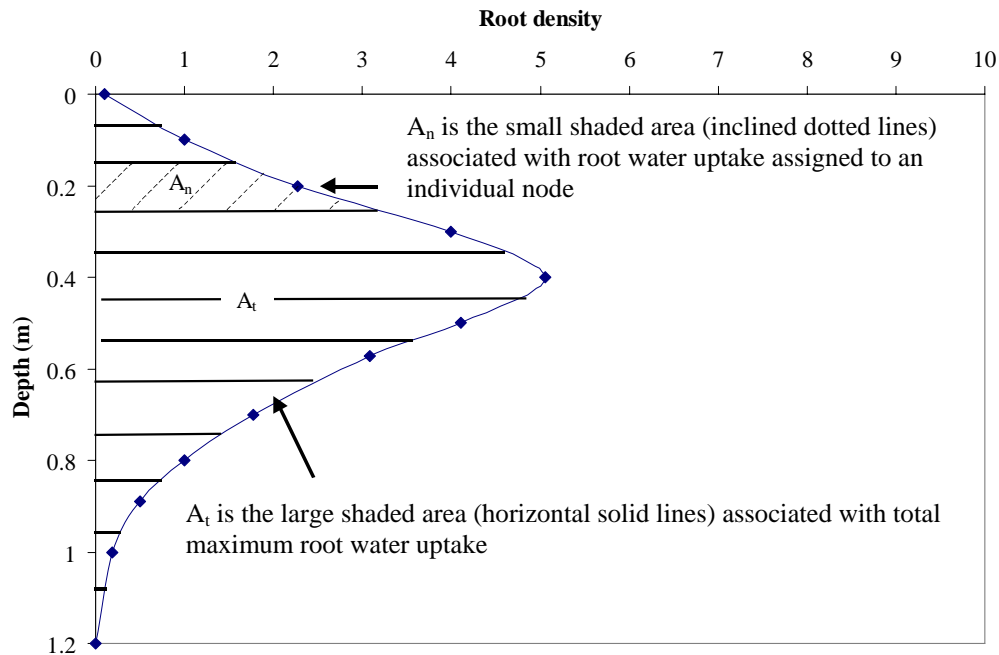


Figure III.3: Root distribution and water uptake

Referring to Figure III.3, root density (x-axis) is the number assigned to each node (values provided in “Water Uptake Distribution”); when plotted versus depth, this results in a root distribution. Integrating this shape gives the total area A_t , which is internally normalized by HYDRUS to be equal to one. The area is subsequently discretized into sub-areas (A_n) associated with each individual node. The root water uptake assigned to each node is:

$$\text{Nodal water uptake} = \text{total transpiration} \times A_n / A_t$$

III.3 Root Distribution Functions

Another way of defining a root distribution is by assigning root distribution parameters that belong to the following root distribution function built into HYDRUS (Vrugt et al., 2001):

$$\beta(z) = \frac{1}{\lambda} \left[1 - \frac{z}{z_m} \right] e^{-\frac{1}{\lambda} \frac{P_z}{z_m} |z^* - z|} \quad (\text{III.2})$$

$$\beta(z^*) = \frac{1}{\lambda} \left[1 - \frac{z^*}{z_m} \right] \quad (\text{III.3})$$

where $\beta(z)$ denotes the dimensionless spatial root distribution as a function of depth z ; z_m is the maximum rooting depth; λ , P_z (parameter A in HYDRUS), and z^* (depth of maximum potential uptake) are fitting parameters. The value of P_z is set to unity for $z > z^*$. Figures III.4 and III.5 demonstrate the effects of varying z^* and P_z for the case where $\lambda=1$ and $z_m=1$. When $\lambda=1$, the root density at depth $z^*=1-z/z_m$ (see Figures III.4 and III.5). For $z > z^*$, the root distribution decays exponentially. The parameter P_z controls the root density at depths $z < z^*$.

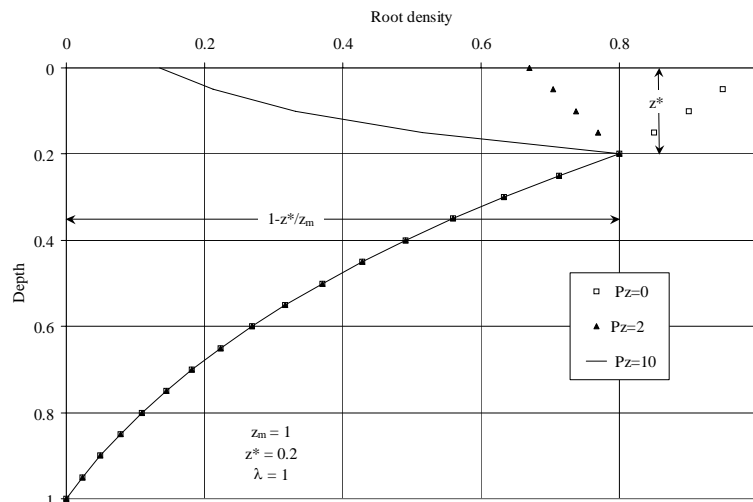


Figure III.4. Effect of P_z on root distribution for $\lambda=1$, $z_m=1$, and $z^*=0.2$

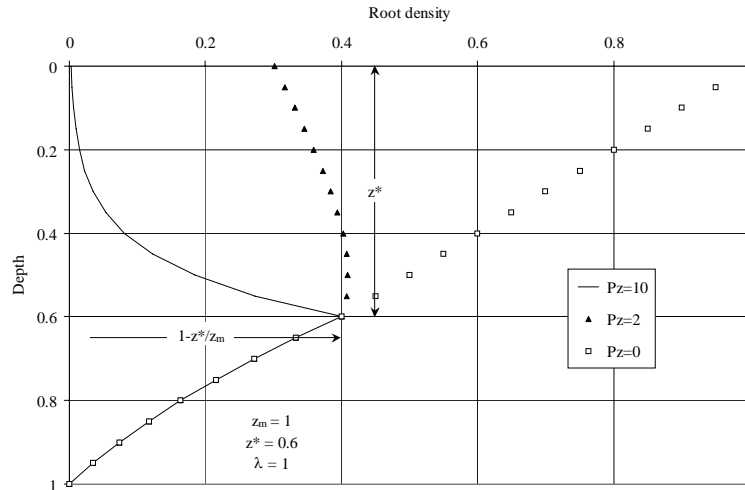


Figure III.5. Effect of P_z on root distribution for $\lambda=1$, $z_m=1$, and $z^*=0.6$

Figure III.6 shows the effect of the parameter λ , which scales the root density at depth z^* (see Equation III.3). The effect of parameter P_z remains restricted to depths $z < z^*$ as was demonstrated in Figures III.4 and III.5.

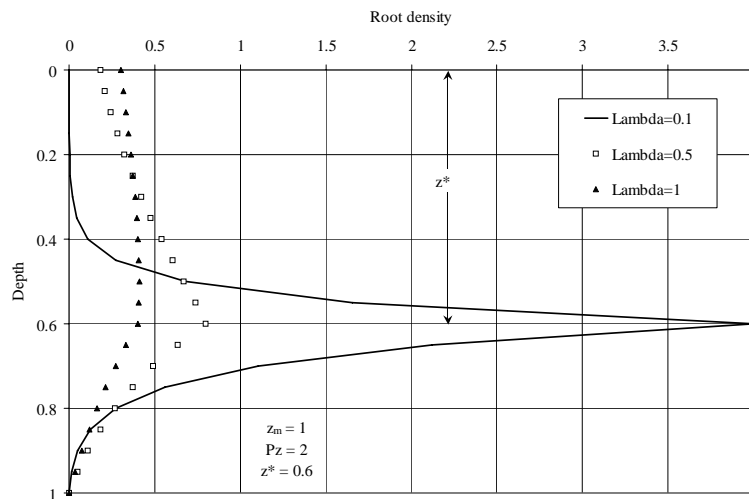


Figure III.6. Effect of λ on root distribution for $z_m=1$ and $z^*=0.6$

2-Dimensional Root Distribution

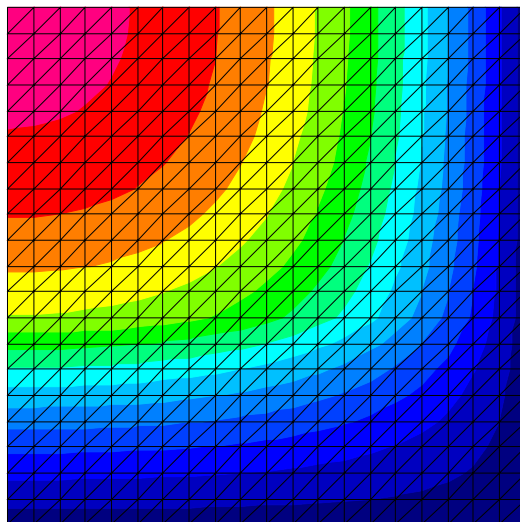
A radial root distribution may be implemented using a model similar to that depicted by Equation II.1 (Vrugt et al., 2001):

$$\beta(r) = \left[1 - \frac{r}{r_m} \right] e^{-\frac{P_r}{r_m} |r^* - r|} \tag{III.4}$$

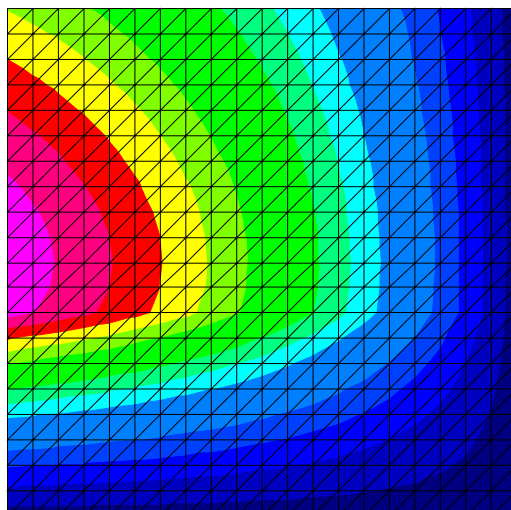
where δ , r_m and P_r have the same meaning in the radial (r) direction as λ , z_m and P_z in Equation II.1.

The 2-dimensional root distribution $\beta(r,z)$ is now simply the product of Equations III.2 and III.4.

When the check box “Horizontal Distribution” is selected as shown below, the user is prompted to enter the required input parameters for a horizontal (radial) root distribution. The figures below show the result of varying the parameters A and Delta (or P_z and δ).



Root Distribution Parameters	
Parameters for Vertical Distribution	
Maximum Rooting Depth	100
Depth of Maximum Intensity	60
Parameter λ	1
Parameter Δ	1
<input checked="" type="checkbox"/> Horizontal Distribution	
Parameters for Horizontal Distribution	
Maximum Rooting Radius	100
Radius of Maximum Intensity	100
Parameter Δ	1
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	



Root Distribution Parameters	
Parameters for Vertical Distribution	
Maximum Rooting Depth	100
Depth of Maximum Intensity	60
Parameter λ	1
Parameter Δ	2
<input checked="" type="checkbox"/> Horizontal Distribution	
Parameters for Horizontal Distribution	
Maximum Rooting Radius	100
Radius of Maximum Intensity	100
Parameter Δ	2
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

APPENDIX IV

SCALING FACTORS

IV.1 Background

HYDRUS-2D implements a scaling procedure designed to simplify the description of spatial variability of the unsaturated soil hydraulic properties in the flow domain. The code assumes that the hydraulic variability in a given area can be approximated by means of a set of linear scaling transformations that relate the individual soil hydraulic characteristics $\theta(h)$ and $K(h)$ to reference characteristics $\theta^*(h^*)$ and $K^*(h^*)$. The technique is based on the similar media concept introduced by Miller and Miller [1956].

Miller and Miller (1956) stated that it is possible to obtain detailed similitude of interface shapes and microscopic flow patterns between two media whose solid geometries differ only by a constant magnifying factor. Two such media are called ‘similar’ media, by analogy to the familiar term ‘similar’ triangles. A pair of similar media in similar states is illustrated in Figure IV.1. When each of these geometries is reduced, i.e., expressed in terms of a characteristic length, λ , the resulting reduced geometries are identical. The two similar media shown in Figure IV-1 differ only in the scale of their internal microscopic geometries; they have equal porosities.

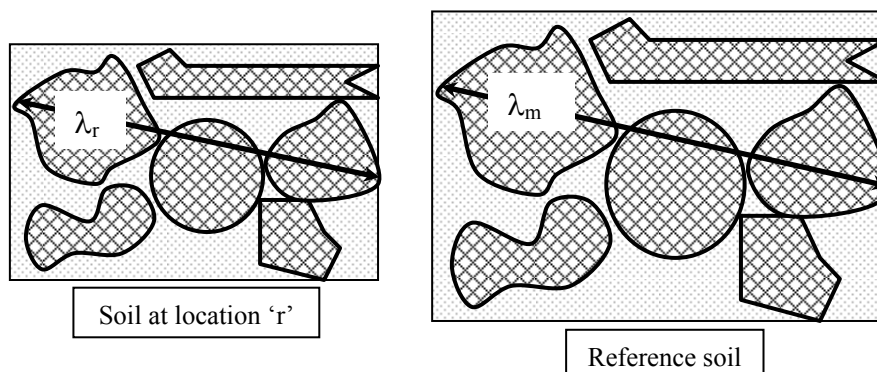


Figure IV.1: Illustration of two similar media

The similar media concept allows results, either experimental or computed, of soil water behaviour in one soil to be used to describe the behaviour in another by employing reduced variables defined in terms of the macroscopic characteristic length. The purpose of scaling is to simplify the description of statistical variations in soil hydraulic properties. By using this simplification, the pattern of spatial variation is described by means of a set of scale factors α_r relating the soil hydraulic properties at each location r to a representative mean or reference soil (Hopmans, 1987). The scaling factor is defined as:

$$\alpha_r = \frac{\lambda_r}{\lambda_m} \quad (\text{IV.1})$$

where $r= 1, 2, \dots, R$ denotes location. The soil water retention and hydraulic conductivity functions at given water contents at any location r are related to a mean or reference h_m and K_m such that for each pressure head:

$$h_r = \frac{h_m}{\alpha_r} \quad (IV.2)$$

and each hydraulic conductivity:

$$K_r = K_m \alpha_r^2 \quad (IV.3)$$

Since different soils generally do not have identical porosity values, h and K are written in terms of degree of saturation S .

Hopmans (1987) reviewed in detail various methods of scaling. This is a demonstration of his Method II, which is derived from Peck et al. (1977). To evaluate the scaling factor at any location, we need to calculate the mean values for both the pressure head and conductivity. These values are given by:

$$h_m(S_i) = R \left\{ \sum_{r=1}^R \left[\frac{1}{h_r(S_i)} \right] \right\}^{-1} \quad (IV.4)$$

and

$$K_m(S_i) = \frac{1}{R^2} \left[\sum_{r=1}^R \sqrt{K_r(S_i)} \right]^2 \quad (IV.5)$$

Equations (IV.4 and IV.5) evaluate the average value (pressure head or conductivity) over all locations r , for any saturation level (S_i). The scaling factors are then calculated as follows:

$$\alpha_r^h = \frac{1}{SL} \sum_{i=1}^{SL} \frac{h_m(S_i)}{h_r(S_i)} \quad (IV.6)$$

$$\alpha_r^k = \sqrt{\frac{1}{SL} \sum_{i=1}^{SL} \frac{K_r(S_i)}{K_m(S_i)}} \quad (IV.7)$$

where SL denotes the different pressure steps at location r ; the scaling factors are averages over all levels of saturation (SL s).

Vogel et al. (1991) demonstrated several cases where the concept of scaling may be utilised in numerical modelling applications. We summarise here one of their exercises:

1. 20 undisturbed soil cores were obtained from the field.
2. Cumulative outflow curves were obtained during pressure desorption.
3. 20 inverse modelling runs were carried out for each of the outflow curves to obtain the soil hydraulic functions of each soil.

Alternatively:

1. Scale the outflow curves and obtain a set of scaling parameters for each sample.
2. Conduct 1 inverse modelling run for the reference outflow curve to obtain the soil hydraulic functions of the reference soil.
3. Use the scaling factors to obtain the soil hydraulic functions of each soil.

Results using both techniques compared relatively well. The latter approach is a time saver since inverse numerical runs are iterative and can be very time-consuming.

IV.2 Stochastic Scaling Factors in HYDRUS

HYDRUS has an option for assigning stochastic scaling factors. This option can be used to:

- Implement the above-described scaling concepts in a random manner.
- Assign random hydraulic conductivities to the domain. The conductivity of each node will then be multiplied by the scaling factor, which randomly varies from node to node, thus resulting in a grid with randomly variable conductivities.

The stochastic distribution option requires three input parameters for each scaling factor:

1. Standard deviation: Indicates the extent of variation of the scaling factors; the higher this value the wider the range over which the scaling factors vary (larger difference between minimum and maximum values).
2. Correlation length in x-direction: A number that indicates the manner in which the scaling factor changes in the x-direction.
3. Correlation length in z-direction: As above for the z-direction.

The meaning of correlation length:

The measured values of any variable are correlated up to a certain distance. This means that values close to a given point have a large probability to have similar values. The correlation length indicates the distance to where those values are fairly similar, or correlated. The correlation length can be different in different directions (x-horizontal and z-vertical). For example, one would expect that the correlation length in the vertical direction to be relatively small because of layering, while it could be much larger in the horizontal direction. The correlation length can be different for different properties.

The correlation length may simply be defined as “the distance over which a significant correlation exists (Kutilek and Nielsen, 1994).

The correlation length L in a 1-dimensional transect (x or z) is commonly defined as:

$$r = r_0 e^{-\frac{x}{L}} \quad (IV.8)$$

where r_0 represents the base correlation coefficient taken equal to 1, which means that at the measured point itself ($x=0$), the correlation coefficient is equal to 1 (scaling factor multiplier=1). At a distance equal to the “Correlation Length, L ”, r is equal to e^{-1} (0.3678). Figure IV.2 shows the manner in which L affects the correlation coefficient (and hence the scaling factors). Note that the dotted line intersects each of the three curves at a distance equal to L .

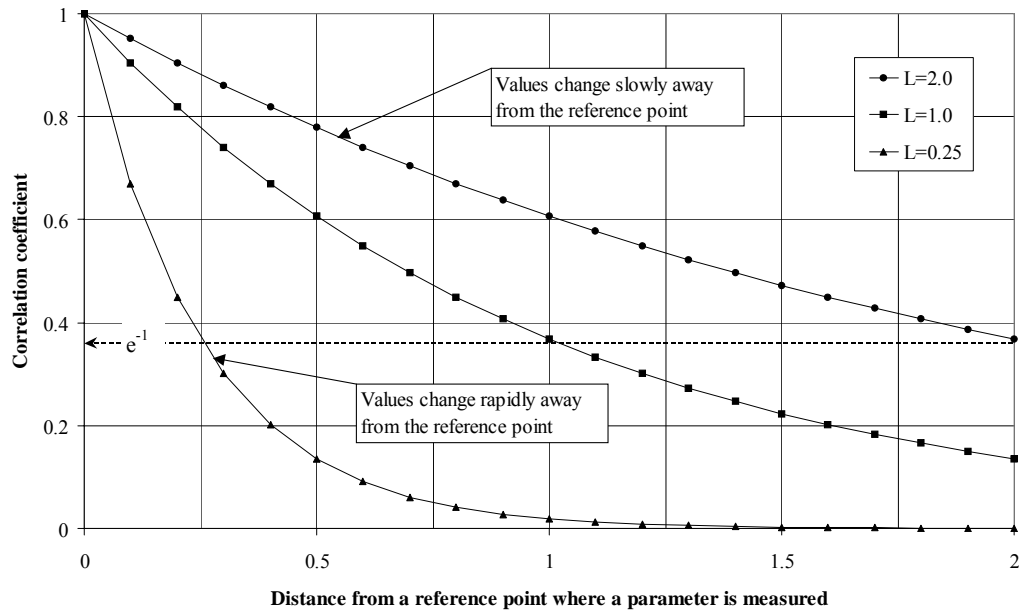


Figure IV.2: The concept of Correlation Length

Effect of standard deviation on distribution of scaling factors:

The standard deviation affects the extent of variation in the scaling factors, where higher deviations lead to more variation in the scaling factors. The figures below demonstrate the impact of a 10-fold increase in the standard deviation. Note that when the standard deviation is increased, the range of scaling factors follows course (see range of colour palette bars in Figure IV.3; 0-4 on the right as compared to 0.6-1.3 on the left).

Effect of correlation length on distribution of scaling factors:

The correlation length (L) is a measure of how similar the scaling factors are in a particular direction. A high L_x means that the scaling factor maintains its value for a long horizontal distance away from the reference point, that is, it does not change rapidly. Conversely, a low L_x means that it changes at a small horizontal distance.

Figure IV.4 shows the effect of different L values in the x - and z -directions. When $L_x/L_z=0.1$, the contours are vertically stretched, and when $L_x/L_z=10$, the contours are horizontally stretched.

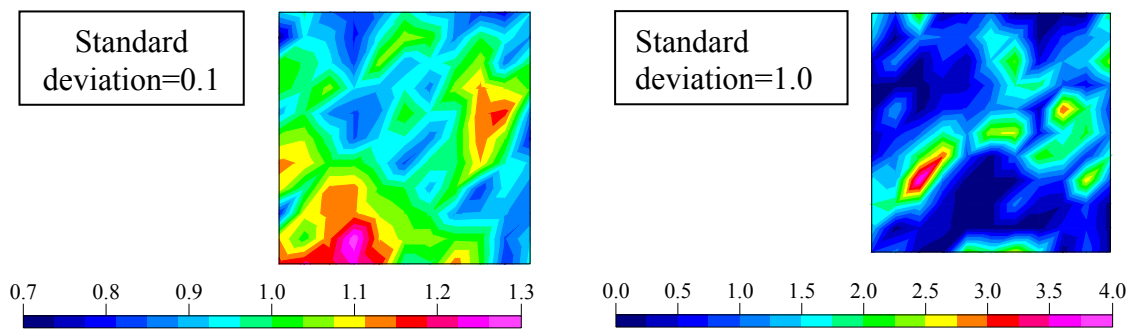


Figure IV.3: Effect of standard deviation on distribution of scaling factors

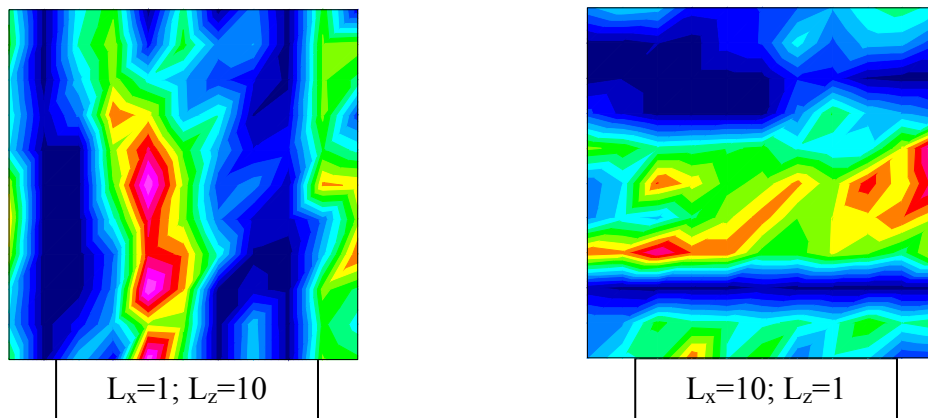


Figure IV.4: Effect of correlation length on distribution of scaling factors

APPENDIX V

INVERSE SOLUTION

This process is usually referred to as “Model calibration”, or “Inverse parameter estimation”. The technique is used in many applications, some of which are listed in the Table V.1.

Table V.1: Example applications for inverse solution

Experiment/application	Calibration data	Goal (Key Prediction)
Field drainage	Cumulative flow, pressure heads	Field-scale K_{sat}
Disc infiltrometer	Cumulative inflow, pressure heads, water contents	Water retention and hydraulic conductivity parameters
Cultivated land	Neutron probe data (water contents)	Leakage (recharge to water table)
Field tracer experiment	Solute concentration in injection/capture wells	K_{sat} , flow rates

It is crucial that a correct conceptualisation of the problem be established first (involving boundary conditions and initial conditions); otherwise the inverse solution will not yield a result or may yield physically unrealistic parameters. It is the modeller’s responsibility to distinguish between “realistic” and “unrealistic” parameters.

V.1 Inverse Method

The ultimate aim of model calibration is to adjust input parameters such that the model predictions fit observations in an optimal manner. The so-called “best fit” is assessed objectively by using a quantitative measure of the quality of the fit by measuring the errors involved. Minimising the sum of the squares of the residuals is the most common procedure and is adopted in HYDRUS:

$$S_r = \sum_{i=1}^n (q_{p,i} - q_{o,i})^2 = \sum_{i=1}^n R_i^2 \quad (V.1)$$

where n is the number of calibration data points (e.g., measured water contents), $q_{p,i}$ is the i^{th} predicted value, and $q_{o,i}$ is the i^{th} observed value. The difference between predicted and observed values in Equation V.1 is called the residual R_i (see Figure V.1). Each point in a calibration data set can be assigned a weighting factor depending upon the degree of confidence or the significance associated with individual calibration points (provided by the user; w_{ij}). The weighting factors give individual points higher or lower significance (or weight) relative to the total sum S_r . The residuals are then called “weighted residuals”. HYDRUS implements another weighting factor v_j for each data set to allow for differences in absolute values of various data sets, such as pressure heads and water contents. Heads are measured in cm and hence might have higher values when compared to volumetric water content.

Pressure head contributions to S_r are generally hence much higher than water contents, which may lead to a biased calibration.

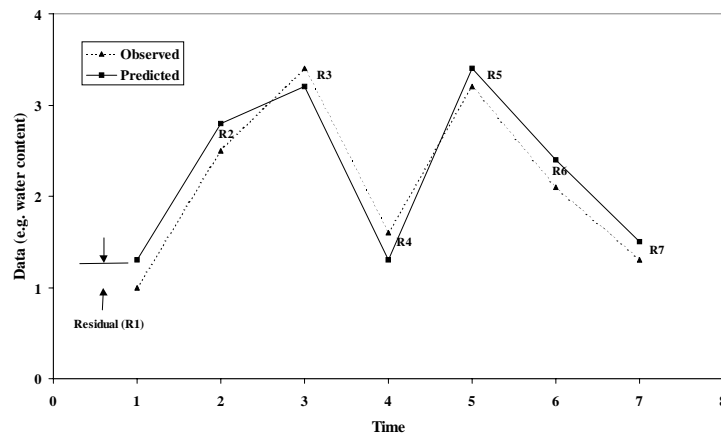


Figure V.1. Meaning of residuals

Another contribution to bias is the use of different unit systems; for example, the hydraulic conductivity could have a very low value when expressed in m/s but be thousands of times higher when expressed in cm/day. HYDRUS overcomes this problem by introducing a weighting factor associated with each data type. The term representing the sum of the weighted residuals from various data types hence becomes:

$$S_{wr} = \sum_{j=1}^m v_j \sum_{i=1}^n w_{i,j} R_i^2 \quad (V.2)$$

where m is the number of data types used, v_j is the weighting factor associated with a data type provided by HYDRUS, and w_{ij} is the weighting factor associated with individual data points provided by the user. The definition of the weighting factors v_j and $w_{i,j}$ is found in the HYDRUS Technical Manual (p.72).

Differences between measured and predicted soil hydraulic parameters (such as retention parameters and the saturated hydraulic conductivity), and having known hydraulic parameters (also referred to as prior information) form another two terms that look like Equation V.2. The sum of the three terms makes what is known as the Objective Function (see HYDRUS Technical Manual, p.71, Eq. 7.1). HYDRUS uses the Marquardt-Levenberg optimisation algorithm to minimise the objective function, that is, to come up with parameters that produce the minimum difference between the observed and predicted values.

V.2 Statistical Issues

V.2.1 Distribution of Residuals

Figure V.2 shows an ideal case where the distribution of the weighted residuals is random and centred around a zero value.

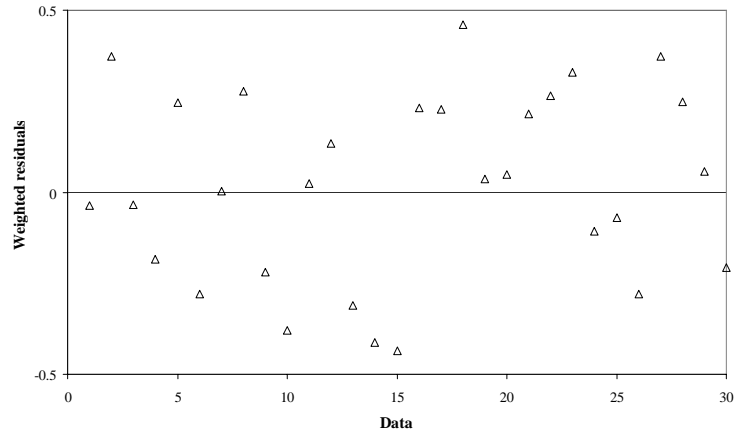


Figure V.2. Random distribution of weighted residuals

By contrast, Figure V.3 shows a case where the weighting process is inadequate. The data between 0 and 16 are under-weighted; they may represent concentration data that have low values, whereas data in the range 20-30 may represent much higher pressure head values.

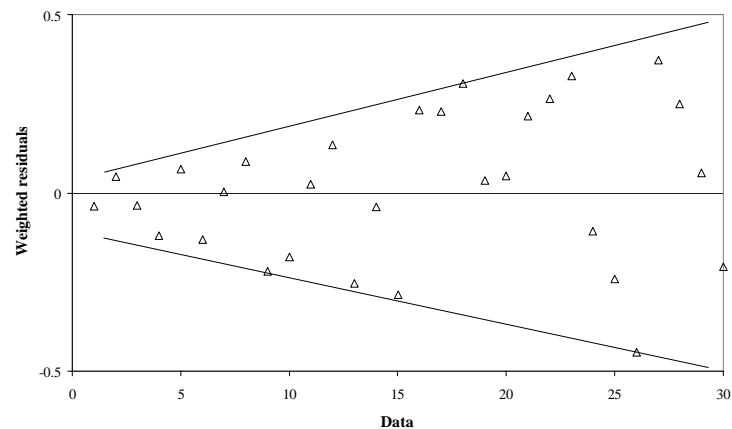


Figure V.3. Distribution of weighted residuals suggesting inadequate weighting

Figure V.4 shows another case with a systematic bias in the distribution. This case may indicate an erroneous conceptualisation of the problem (e.g. an incorrect boundary condition).

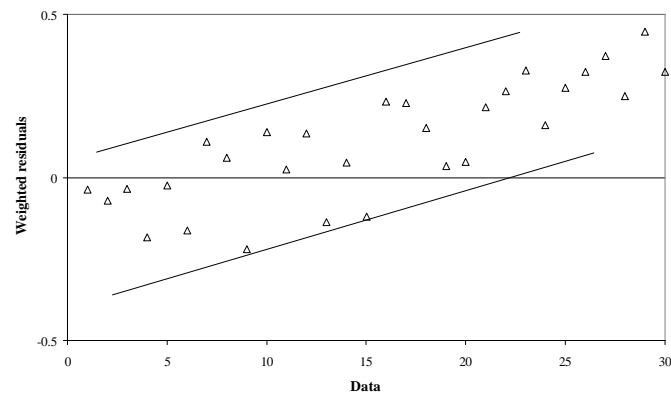


Figure V.4. Distribution of weighted residuals with systematic bias

V.2.2 Confidence Limits

The lower and upper bounds of the confidence limits indicate the sensitivity level of a model to any particular parameter measured by the response of the objective function. Figure V.5 shows a case where the model is very sensitive to a parameter in that a small change in the parameter values causes a dramatic change in the objective function.

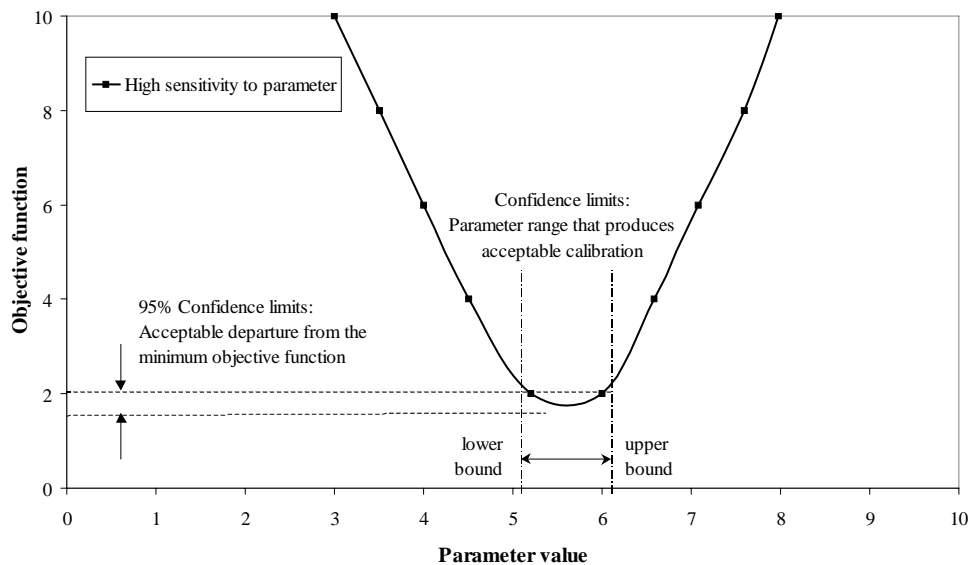


Figure V.5: Model with high sensitivity to parameter change

On the other hand, Figure V.6 shows a model with very low sensitivity to a particular parameter. Significant changes in the parameter value now cause only a very small change in the objective function. In such cases one may consider eliminating the parameter from the optimisation scheme.

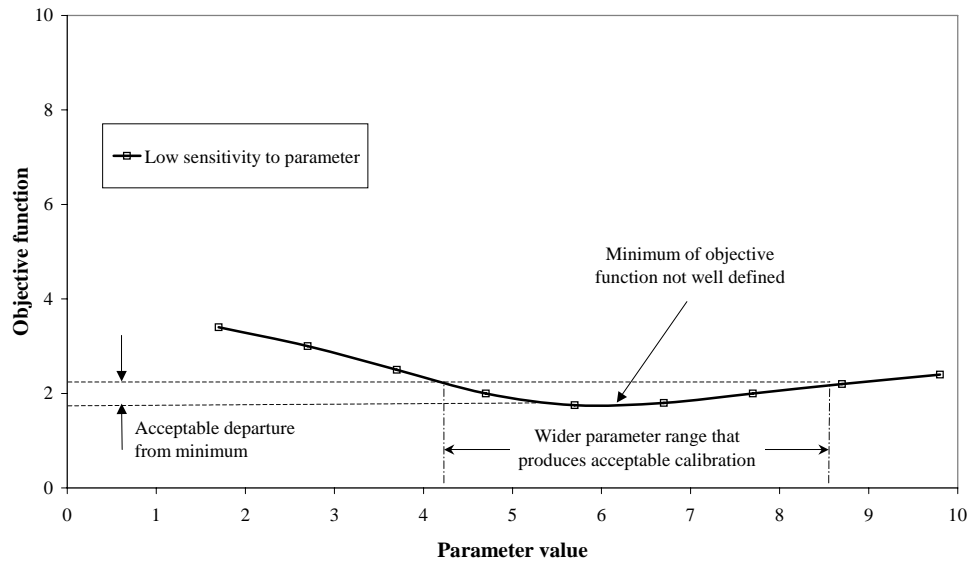


Figure V.6: Model with low sensitivity to parameter change

V.2.3 Goodness of Fit and Errors

The goodness of fit is measured using the r^2 values. The equation for r^2 is found in the HYDRUS Technical Manual (p.73, Eq. 7.3). The r^2 indicates that the best fit is obtained when its value approaches or is equal to 1. However, an r^2 close to 1 does not always guarantee a good fit. Users should always investigate this problem. It is best to always plot the observed data along with the model predictions for a visual inspection. Figure V.7 demonstrates a case where two regressions have r^2 values of unity, but one of them has the wrong slope; note that only the 1:1 (45°) line represents a perfect fit. This line is referred to as the “Line of Perfect Fit”.

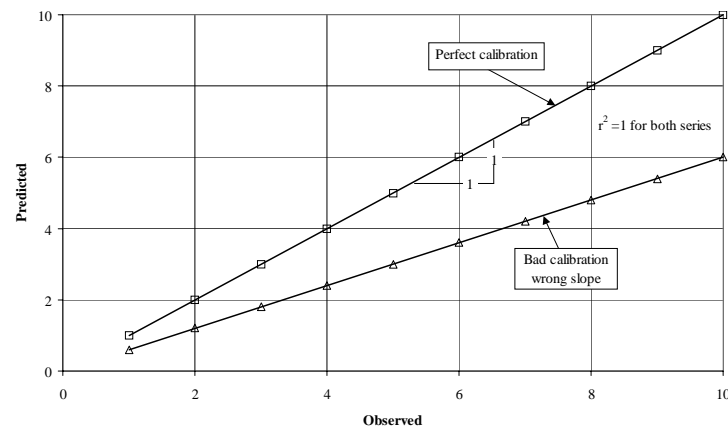


Figure V.7: Goodness of fit

The standard error of the estimated parameter quantifies the spread of the model predictions around the observed values, and is defined as follows:

$$s(b_j) = \sqrt{\frac{S_r}{n - n_p}} \quad (\text{V.3})$$

where n_p is the number of unknown parameters to be fitted. The T-value provides an absolute measure for the deviation:

$$T = \frac{b_j}{s(b_j)} \quad (\text{V.4})$$

where b_j is the final estimate of the parameter.

V.2.4 Correlation Matrix

HYDRUS produces a correlation matrix, which specifies the degree of correlation between the fitted parameters. The correlation matrix quantifies changes in model predictions caused by small changes in the final estimate of a particular parameter, relative to similar changes as a result of changes in the other parameter (HYDRUS Technical Manual). The matrix as reported in the “Fit.out” output files, is shown in Table V.2:

Table V.2: Correlation matrix

	Alpha	n	Cond
Alpha	1		
n	-0.216	1	
Cond	0.98		1

The correlation matrix indicates whether the parameters are correlated or not, where 1 indicates perfect positive correlation and -1 indicates perfect negative correlation. The matrix shown above indicates that there is a significant correlation only between Alpha and the saturated conductivity. The matrix diagonal has always values of unity because it refers to correlation with the parameter itself. Figure V.8 shows an example of two highly correlated parameters; notice that a straight line was fitted to the data with an R^2 of 0.952.

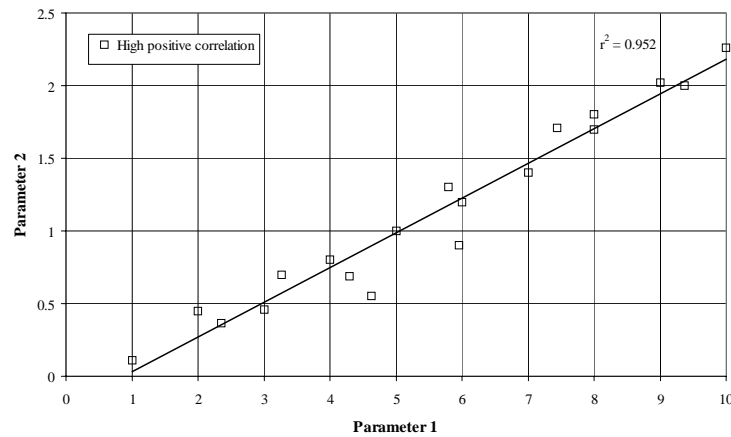
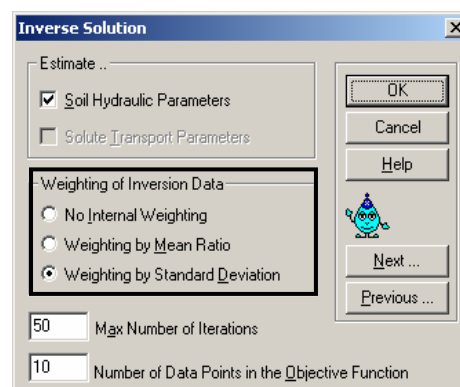


Figure V.8: Parameters correlation

Correlated parameters are often closely associated with non-unique solutions. More than one set of parameters may then produce a good fit to the observed trends. This phenomenon results in a high degree of predictive uncertainty. In cases where a high degree of correlation does exist, the number of optimised parameters must be decreased, or new types of calibration data must be added. For example, if you are only monitoring pressure heads, try to also include flow rates. We emphasize that increasing the number of calibration data in the set that produced the highly correlated parameters by itself may not solve the problem.

V.2.5 Weighting Parameters

The number of the calibration data, and their absolute values, can have a profound impact on the objective function. For example, if you use time units as hours, the absolute value of the hydraulic conductivity will be 60 times higher compared to when a time unit of minutes is used. The absolute difference between the observed and predicted values hence will be higher also, and thus will have a greater impact on the objective function (the sum of all differences). This means that you will get different results depending upon what unit is used, which seems illogical. HYDRUS uses two approaches to manage this problem: weighting by mean ratio and weighting by standard deviation. For more details, refer to the HYDRUS Technical Manual P. 72.



We demonstrate how weighting factors avoid the problem introduced by changing the units. Let us copy project 'Inver-b5' and name this project 'Weight' (CD.\HYDRUS Manual Projects\Appendix). We change the time units to hours and re-run the project.

The following information is copied from the output file "Fit.out". Refer to Table V.3 (the values under column 'Weight'). Notice the high weighting factors for data points 8 and 9. These are hydraulic conductivity measurements in cm/minute (time units is minutes).

Table V.3: Inverse data from Project 'Inver-b5'

Obs	Time	QUANTITY	Type	Position	Weight	
1	33	0.179	2	1	7.020282	
2	66	0.211	2	1	7.020282	
3	100	0.246	2	1	7.020282	
4	133	0.271	2	1	7.020282	
5	166	0.282	2	1	7.020282	
6	200	0.286	2	1	7.020282	
7	200	-46	1	1	2	
8	-92	3.35E-04	6	1	2786.627	0.93352
9	-52	1.35E-03	6	1	2786.627	3.761946
10	-306	0.132	5	1	14.04056	

The data in Table V.4 are from the same project but with time unit system changed to hours (absolute value of conductivity increased 60 folds). Notice that the weighting factors for data points 8 and 9 have decreased by exactly 60 fold. The numbers in bold are the products of the measured value and the weighting factor; notice that they are equal to the corresponding numbers in Table V.3. These are the numbers that will be incorporated into the objective function. All other weighting factors have remained the same since they are independent of time (the unit we changed).

Table V.4: Inverse data from Project 'Inver-b5'

Obs	Time	QUANTITY	Type	Position	Weight	
1	0.55	0.179	2	1	7.020282	
2	1.1	0.211	2	1	7.020282	
3	1.667	0.246	2	1	7.020282	
4	2.217	0.271	2	1	7.020282	
5	2.767	0.282	2	1	7.020282	
6	3.34	0.286	2	1	7.020282	
7	3.34	-46	1	1	2	
8	-92	0.02	6	1	46.44379	0.928876
9	-52	0.081	6	1	46.44379	3.761947
10	-306	0.132	5	1	14.04056	

V.3 Intelligent Data Acquisition

It is extremely important that the calibration data be as representative as possible of any spatial and temporal variations in a model proposed for calibration. Calibration data are either obtained from laboratory or field experiments. A sound conceptualisation and physical understanding of the system should precede the instrumentation stage. Preliminary predictive modelling enables a modeller to identify what type of data are needed (pressure heads, water contents, etc.), the location where they are mostly needed (spatial distribution), and the frequency at which they should be acquired (temporal distribution). Unless this process is carefully undertaken, one might either end up with redundant data, or have data gaps in space and/or time that could result in a biased calibration. The following simulation will shed some light on how to obtain a useful calibration data set when simulating a drainage problem.

Project ‘Dr-evap’ (CD.\HYDRUS Manual Projects\Appendix) involves a fine-textured soil block 10-m wide and 1-m deep that drains into a ditch in two consecutive stages in the absence and presence of evaporation, respectively. The pressure head in the saturated zone is measured, and the negative pressure head in the unsaturated zone is monitored at a depth of 10 cm.

The predictive modelling process reveals the following:

1. Figure V.9 shows that during drainage alone (day 1), pressure head changes are less significant away from the drain. Hence, observation wells should be placed with a higher density close to the drain.
2. Figure V.9 shows that 1 m from the drain an almost instantaneous decrease of about 15 cm occurs in the water table level. This suggests having a higher sampling frequency at this location during this time period.
3. Figure V.9 shows that in the presence of evaporation (rate=1 cm/day), the water table behaves significantly differently (water table will decline away from the drain at almost the same rate as close to it). This means that the dominant processes are different in high-evaporation, arid areas.

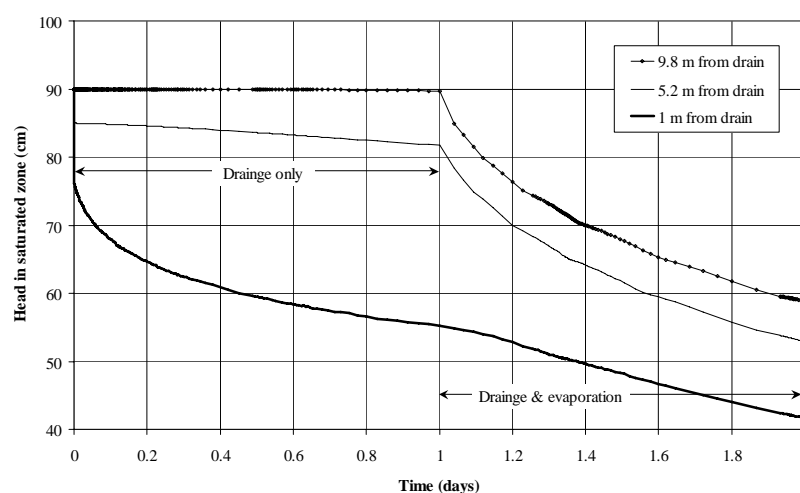


Figure V.9: Pressure head data from the saturated zone at various distances from the drain

4. Figure V.10 shows that during drainage alone, monitoring pressure heads (with tensiometers) in the unsaturated zone alone does not portray the full picture. However, when evaporation is modelled, the data are more valuable.

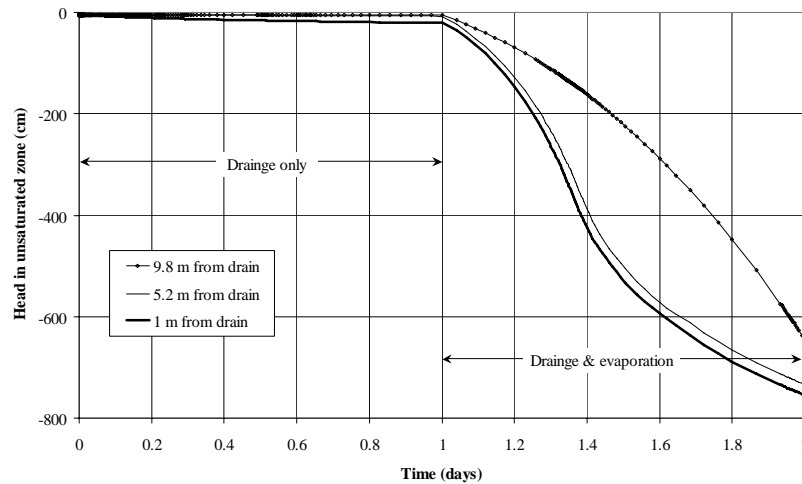


Figure V.10: Head data from the unsaturated zone at various distances from the drain

5. Figure V.11 shows that the cumulative seepage flux neither reflects the subtleties in the groundwater dynamics, nor the pressure heads in the unsaturated zone. Hence, we do not expect to obtain a good calibration from drainage data alone. However, those data may still be valuable when used together with pressure head measurements.

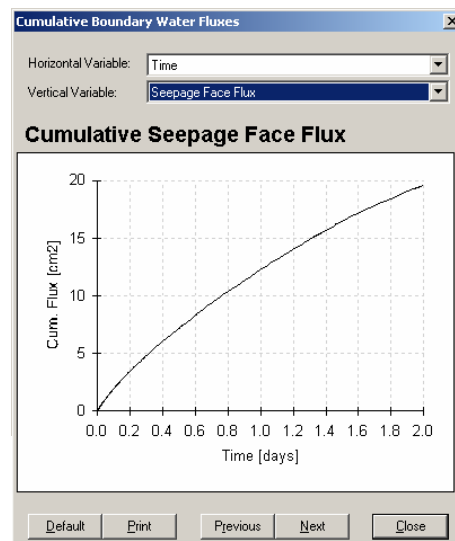


Figure V.11: Cumulative seepage outflow

APPENDIX IV

SCALING FACTORS

IV.1 Background

HYDRUS-2D implements a scaling procedure designed to simplify the description of spatial variability of the unsaturated soil hydraulic properties in the flow domain. The code assumes that the hydraulic variability in a given area can be approximated by means of a set of linear scaling transformations that relate the individual soil hydraulic characteristics $\theta(h)$ and $K(h)$ to reference characteristics $\theta^*(h^*)$ and $K^*(h^*)$. The technique is based on the similar media concept introduced by Miller and Miller [1956].

Miller and Miller (1956) stated that it is possible to obtain detailed similitude of interface shapes and microscopic flow patterns between two media whose solid geometries differ only by a constant magnifying factor. Two such media are called ‘similar’ media, by analogy to the familiar term ‘similar’ triangles. A pair of similar media in similar states is illustrated in Figure IV.1. When each of these geometries is reduced, i.e., expressed in terms of a characteristic length, λ , the resulting reduced geometries are identical. The two similar media shown in Figure IV-1 differ only in the scale of their internal microscopic geometries; they have equal porosities.

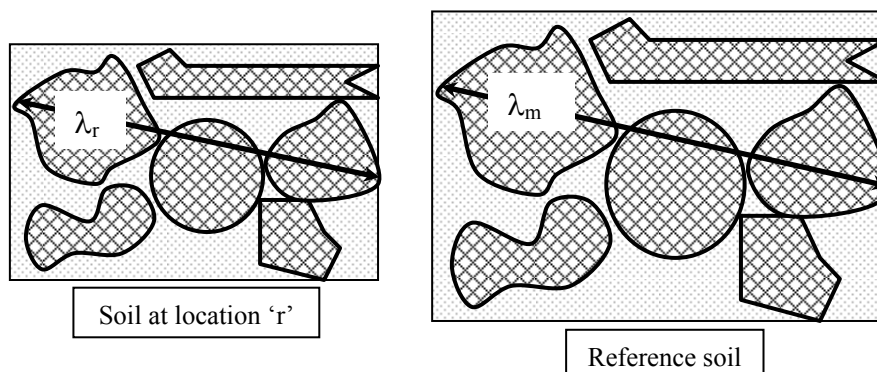


Figure IV.1: Illustration of two similar media

The similar media concept allows results, either experimental or computed, of soil water behaviour in one soil to be used to describe the behaviour in another by employing reduced variables defined in terms of the macroscopic characteristic length. The purpose of scaling is to simplify the description of statistical variations in soil hydraulic properties. By using this simplification, the pattern of spatial variation is described by means of a set of scale factors α_r relating the soil hydraulic properties at each location r to a representative mean or reference soil (Hopmans, 1987). The scaling factor is defined as:

$$\alpha_r = \frac{\lambda_r}{\lambda_m} \quad (\text{IV.1})$$

where $r= 1, 2, \dots, R$ denotes location. The soil water retention and hydraulic conductivity functions at given water contents at any location r are related to a mean or reference h_m and K_m such that for each pressure head:

$$h_r = \frac{h_m}{\alpha_r} \quad (\text{IV.2})$$

and each hydraulic conductivity:

$$K_r = K_m \alpha_r^2 \quad (\text{IV.3})$$

Since different soils generally do not have identical porosity values, h and K are written in terms of degree of saturation S .

Hopmans (1987) reviewed in detail various methods of scaling. This is a demonstration of his Method II, which is derived from Peck et al. (1977). To evaluate the scaling factor at any location, we need to calculate the mean values for both the pressure head and conductivity. These values are given by:

$$h_m(S_i) = R \left\{ \sum_{r=1}^R \left[\frac{1}{h_r(S_i)} \right] \right\}^{-1} \quad (\text{IV.4})$$

and

$$K_m(S_i) = \frac{1}{R^2} \left[\sum_{r=1}^R \sqrt{K_r(S_i)} \right]^2 \quad (\text{IV.5})$$

Equations (IV.4 and IV.5) evaluate the average value (pressure head or conductivity) over all locations r , for any saturation level (S_i). The scaling factors are then calculated as follows:

$$\alpha_r^h = \frac{1}{SL} \sum_{i=1}^{SL} \frac{h_m(S_i)}{h_r(S_i)} \quad (\text{IV.6})$$

$$\alpha_r^k = \sqrt{\frac{1}{SL} \sum_{i=1}^{SL} \frac{K_r(S_i)}{K_m(S_i)}} \quad (\text{IV.7})$$

where SL denotes the different pressure steps at location r ; the scaling factors are averages over all levels of saturation (SL s).

Vogel et al. (1991) demonstrated several cases where the concept of scaling may be utilised in numerical modelling applications. We summarise here one of their exercises:

1. 20 undisturbed soil cores were obtained from the field.
2. Cumulative outflow curves were obtained during pressure desorption.
3. 20 inverse modelling runs were carried out for each of the outflow curves to obtain the soil hydraulic functions of each soil.

Alternatively:

1. Scale the outflow curves and obtain a set of scaling parameters for each sample.
2. Conduct 1 inverse modelling run for the reference outflow curve to obtain the soil hydraulic functions of the reference soil.
3. Use the scaling factors to obtain the soil hydraulic functions of each soil.

Results using both techniques compared relatively well. The latter approach is a time saver since inverse numerical runs are iterative and can be very time-consuming.

IV.2 Stochastic Scaling Factors in HYDRUS

HYDRUS has an option for assigning stochastic scaling factors. This option can be used to:

- Implement the above-described scaling concepts in a random manner.
- Assign random hydraulic conductivities to the domain. The conductivity of each node will then be multiplied by the scaling factor, which randomly varies from node to node, thus resulting in a grid with randomly variable conductivities.

The stochastic distribution option requires three input parameters for each scaling factor:

1. Standard deviation: Indicates the extent of variation of the scaling factors; the higher this value the wider the range over which the scaling factors vary (larger difference between minimum and maximum values).
2. Correlation length in x-direction: A number that indicates the manner in which the scaling factor changes in the x-direction.
3. Correlation length in z-direction: As above for the z-direction.

The meaning of correlation length:

The measured values of any variable are correlated up to a certain distance. This means that values close to a given point have a large probability to have similar values. The correlation length indicates the distance to where those values are fairly similar, or correlated. The correlation length can be different in different directions (x-horizontal and z-vertical). For example, one would expect that the correlation length in the vertical direction to be relatively small because of layering, while it could be much larger in the horizontal direction. The correlation length can be different for different properties.

The correlation length may simply be defined as “the distance over which a significant correlation exists (Kutilek and Nielsen, 1994).

The correlation length L in a 1-dimensional transect (x or z) is commonly defined as:

$$r = r_0 e^{-\frac{x}{L}} \quad (IV.8)$$

where r_0 represents the base correlation coefficient taken equal to 1, which means that at the measured point itself ($x=0$), the correlation coefficient is equal to 1 (scaling factor multiplier=1). At a distance equal to the “Correlation Length, L ”, r is equal to e^{-1} (0.3678). Figure IV.2 shows the manner in which L affects the correlation coefficient (and hence the scaling factors). Note that the dotted line intersects each of the three curves at a distance equal to L .

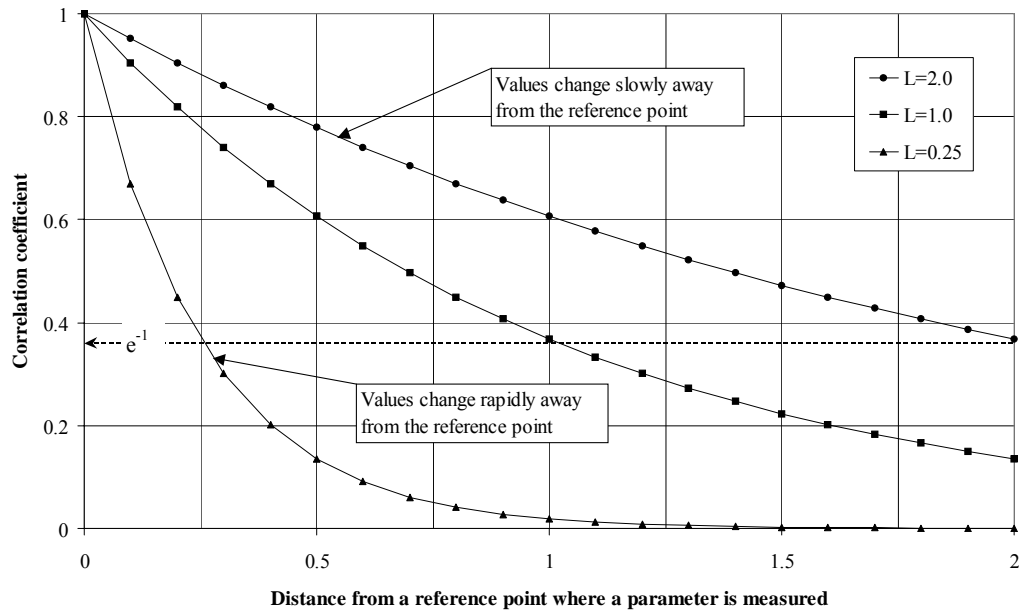


Figure IV.2: The concept of Correlation Length

Effect of standard deviation on distribution of scaling factors:

The standard deviation affects the extent of variation in the scaling factors, where higher deviations lead to more variation in the scaling factors. The figures below demonstrate the impact of a 10-fold increase in the standard deviation. Note that when the standard deviation is increased, the range of scaling factors follows course (see range of colour palette bars in Figure IV.3; 0-4 on the right as compared to 0.6-1.3 on the left).

Effect of correlation length on distribution of scaling factors:

The correlation length (L) is a measure of how similar the scaling factors are in a particular direction. A high L_x means that the scaling factor maintains its value for a long horizontal distance away from the reference point, that is, it does not change rapidly. Conversely, a low L_x means that it changes at a small horizontal distance.

Figure IV.4 shows the effect of different L values in the x - and z -directions. When $L_x/L_z=0.1$, the contours are vertically stretched, and when $L_x/L_z=10$, the contours are horizontally stretched.

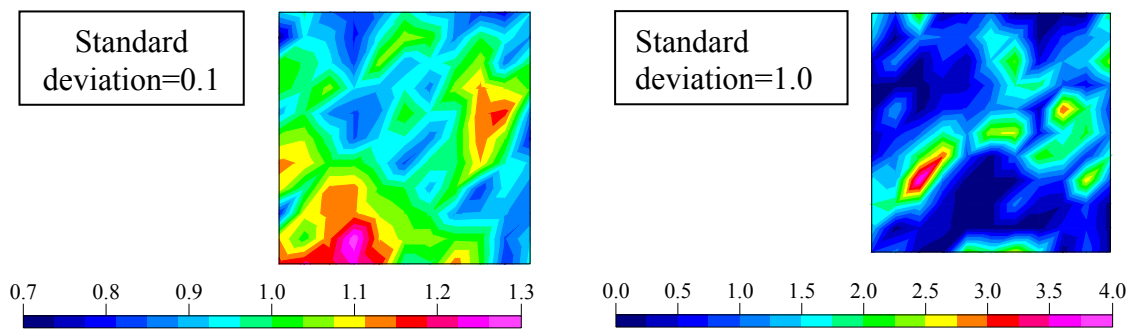


Figure IV.3: Effect of standard deviation on distribution of scaling factors

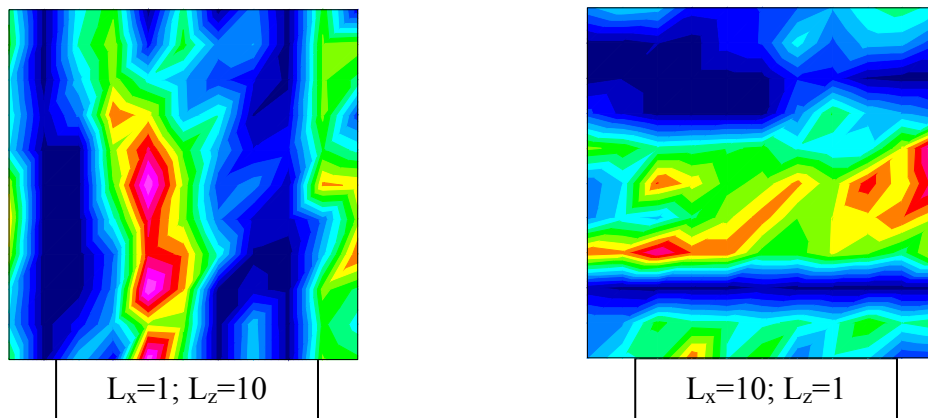



Figure IV.4: Effect of correlation length on distribution of scaling factors

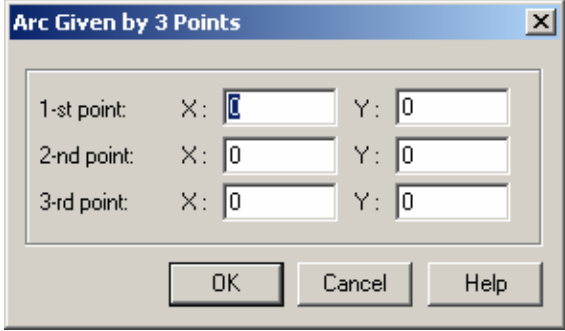
APPENDIX VII

ALPHABETICAL INDEX OF HYDRUS WINDOWS

To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert Arc Numerically 


Function: Inserts an arc numerically.

Required Input: Coordinates of three points on the arc (two end points and one intermediate).



Related pages: N/A

*Display is Conditional: Select 'Tools/Arc Input Mode/Three Points' 

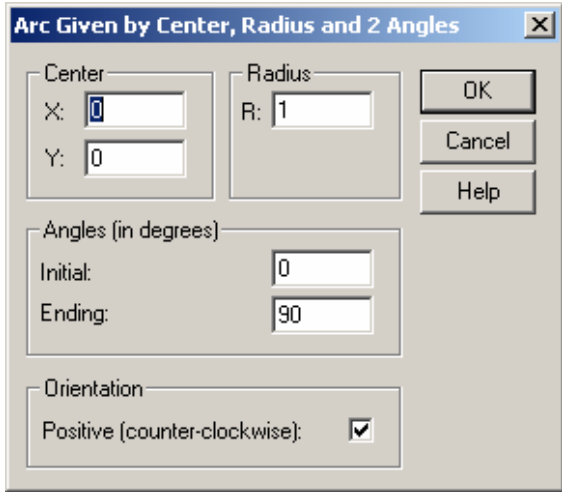
To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert Arc Numerically 


Function: Inserts an arc numerically.


Required Input: Coordinates of centre of arc, radius of arc, starting and ending angles; check box for orientation.

Related pages: 1.17

Related examples: Project 'Grid-1', page 1.16



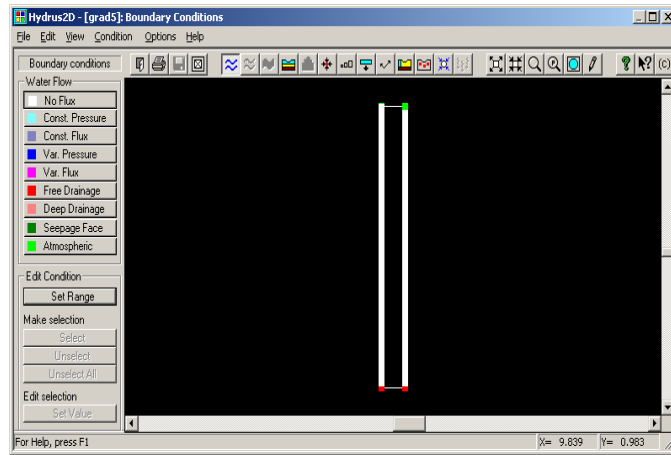
*Display is Conditional: Select 'Tools/Arc Input Mode /Centre, Radius, and Two Angles' 

To display click: Boundary Condition Editor (Boundary) (in Pre-Processing) ()

Function: Sets boundary conditions, material distribution, root distribution, nodal recharge, scaling factors, initial conditions, anisotropy, subregions, and observations nodes.

Required Input: Input is described in individual windows.

Related pages: 1.26 to 1.43 (General); p. xiii; Section 3 and Appendix III (Root distribution); Appendix IV (Scaling factors).

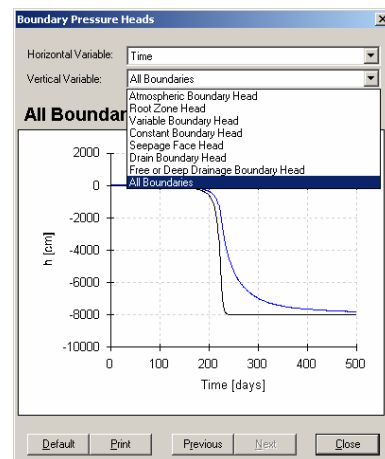


To display click: Pressure Heads (in Post-Processing)

Function: Provides graphical display of average pressure heads across various boundaries.

Required Input: Choice of vertical variable, which represents the relevant boundary type.

Related pages: 1.50; 2.1, 2.2.



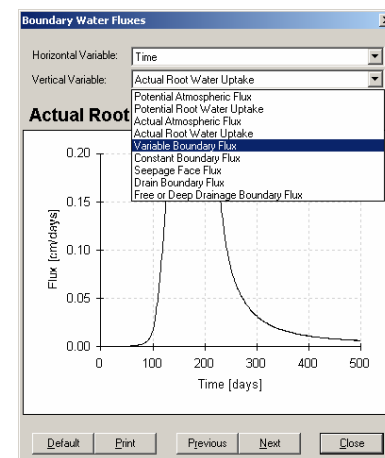
To display click: Water Boundary Fluxes (in Post-Processing)

Function: Provides graphical display of fluxes across various boundaries.

Required Input: Choice of vertical variable, which represents the relevant boundary type.

Related pages: 1.50; 2.1,2.2.

Related examples: Project 'Mbal-2', page 2.8; Project 'Mbal-3', page 2.12. These examples demonstrate the use of the raw flux data from the relevant output files.

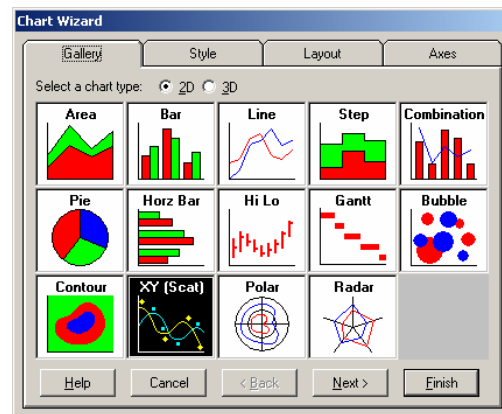



To display click: Graphical Display of Results/Right-click on a chart/Wizard

Function: Provides a variety of charting options.

Required Input: Gallery/Select a chart type; Style/Select style; Layout/change title and footnotes; Axes/changes axes titles.

Related pages: 1.48

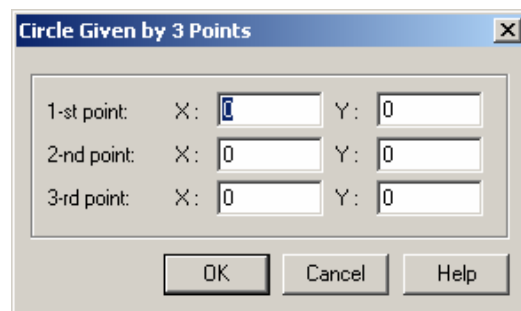



To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert Circle Numerically ()


Function: Inserts a circle.

Required Input: Coordinates of three points on the circle.

Related pages: N/A



*Display is Conditional: Select 'Tools/Circle Input Mode /Three Points' ()

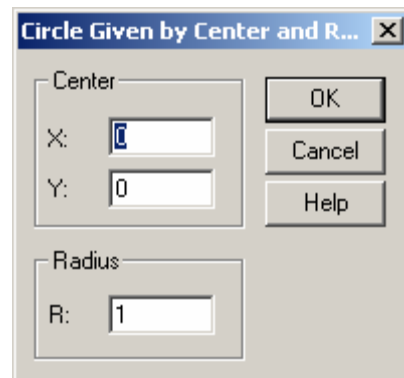
To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert Circle Numerically ()

Function: Inserts a circle.

Required Input: Coordinates of centre and value for radius.

Related pages: 1.24

Related examples: Project 'Grid-3', page 1.22

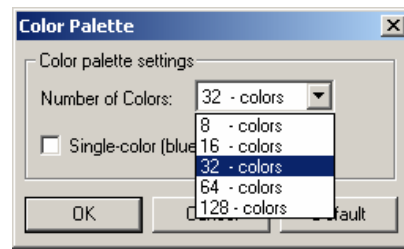


*Display is Conditional: Select 'Tools/Circle Input Mode /Centre and Radius' ()

To display click: Boundary Condition Editor (Boundary)/Options/Colour Palette

Function: Provides options for colour displays; default is 32 colours.

Required Input: Select from drop-down list.

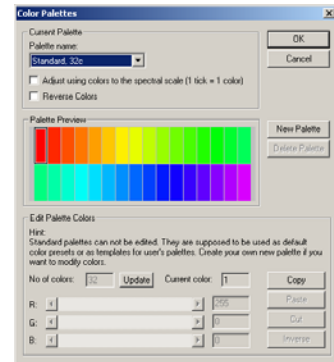


To display click: Graphical Display of Results (Graphics)/Options/Color Palette

Function: Provides colour options to display results.

Required Input: Palette name; it is possible to create custom palettes.

Related pages: 1.49



To display click: Boundary Condition Editor (Boundary)/Condition/Boundary Condition Water Flow/Constant Flux; or LHS Water Flow Buttons/Const. Flux

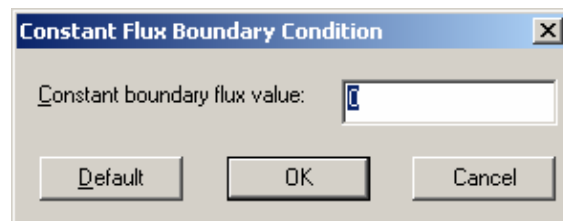
Function: Assigns a constant flux BC for a set of selected boundary nodes.

Required Input: Value of flux; has units of L/T (e.g., cm/day).

Related pages: 1.29

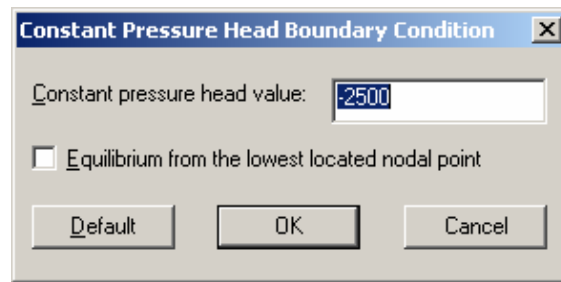
Related examples: Inverse Simulations Series 'a', page 5.4

*Display is Conditional: Select the nodes and left-click when you finish



To display click: Boundary Condition Editor (Boundary)/Condition/ Boundary Condition Water/ Flow/Constant Head; or LHS Water Flow Buttons/Const. Pressure

Function: Assigns a constant pressure head BC for a set of selected boundary nodes.



Required Input: Value of pressure head (positive or negative); has units of L (e.g., cm). When 'Equilibrium from the lowest located nodal point' is checked, the specified value is assigned to the lowest selected point and hydrostatic equilibrium is assumed above that point.

Related pages: 1.27

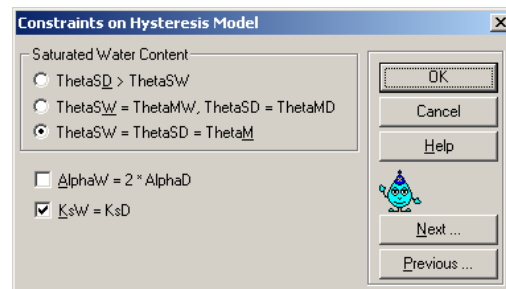
Related examples: Projects 'Pipe-', page 4.12

*Display is Conditional: Select the nodes and left-click when you finish

To display click: Soil Hydraulic Model/Next (in Pre-Processing)

Function: Provides constraints on hysteretic parameters when using the inverse solution.

Required Input: Check appropriate boxes.



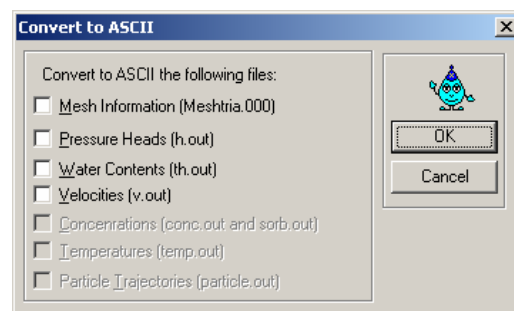
Related pages: 5.2

*Display is Conditional: Select 'Inverse Solution' in 'Main Processes', and 'Hysteresis' in 'Soil Hydraulic Model'.

To display click: Convert Output to ASCII (in Post-Processing)

Function: Converts selected binary output files into ASCII form.

Required Input: Check the appropriate box.



Related pages: 1.53; 2.5

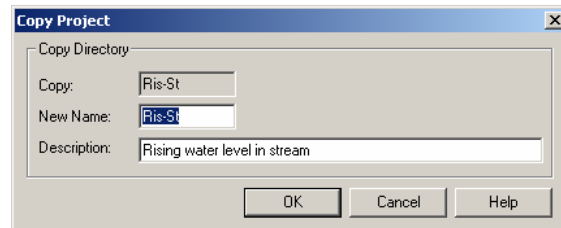
To display click: File/Project Manager/Projects/Copy

Function: Provides copies of HYDRUS projects.

Required Input: A new file name and description. The copy of the project will be saved in the same directory (workspace) by default.

Related pages: 1.2

Related examples: Project 'Dike', page 1.40; Project 'Ris-st', page 4.2



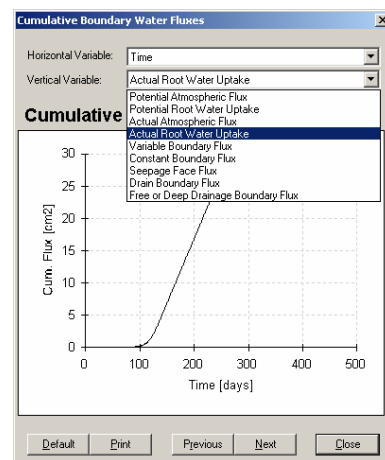
To display click: Cumulative Water Boundary Fluxes (in Post-Processing)

Function: Provides graphical display of cumulative flow volumes across various boundaries.

Required Input: Choice of vertical variable, which represents the relevant boundary type.

Related pages: 1.49; 2.1,2.2.

Related examples: Project 'Mbal-2', page 2.8; Project 'Mbal-3', page 2.12. These examples demonstrate the use of the raw flux data from the relevant output files.

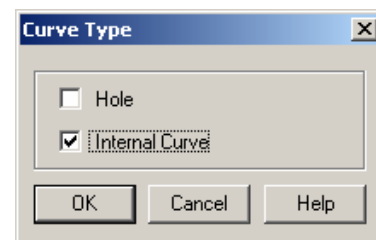


To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/View/Boundary Points/Hole (Side buttons)

Function: Decides whether an internal continuous boundary is a hole (nodes on this boundary represent interface with outside world) or just an internal curve for mesh refinement purposes.

Related pages: 1.24

*Display is Conditional: Select continuous internal boundary.



To display click: Inverse Solution Data (in Pre-Processing)

Function: Provides an entry table for the calibration data.

Required Input: Calibration data, data type, temporal and spatial data details, and weight for individual data points.

	X	Y	Type	Position	Weight
1	33	0.113	2	1	1
2	66	0.1207	2	1	1
3	100	0.142	2	1	1
4	133	0.172	2	1	1
5	166	0.2045	2	1	1
6	200	0.2317	2	1	1
7	233	0.25	2	1	1
8	266	0.2624	2	1	1
9	300	0.272	2	1	1
10	333	0.278	2	1	1
11	366	0.2807	2	1	1
12	400	0.283	2	1	1

Related pages: 5.2, 5.3.

Related examples: All projects in Section 5.

*Display is Conditional: Check the box 'Inverse Solution' in 'Main Processes'

To display click: Right click on any figure in Post-Processing/Edit chart data

Function: Displays the data set for the displayed figure, provides an option to edit the data and change the legend (unless fixed internally).

Required Input: Click on any cell and change its value; click on the column headings and change the legend.

	C1	C2
R663	4.9531	-5140
R664	4.9609	-5150
R665	4.9688	-5150
R666	4.9766	-5160
R667	4.9844	-5170
R668	4.9922	-5180
R669	5	-5190

Related pages: 1.46-1.47

Related examples: Project '1D-Infil' page xvii.

To display click: Boundary Condition Editor (Boundary)/Condition/Boundary Condition Water Flow/Deep Drainage; or LHS Water Flow Buttons/Deep Drainage

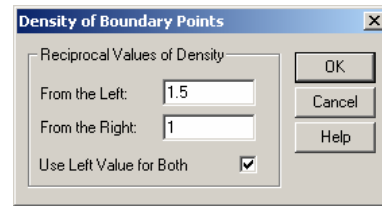
Function: Assigns deep-drainage BC parameters.

Required Input: 'a' and 'b' parameters and reference groundwater level position.

Related pages: 1.31

*Display is Conditional: Select the nodes and left-click when you finish.

To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/View/Boundary Points/Density (side buttons)



Function: Assigns mesh densities on either side of fixed points.

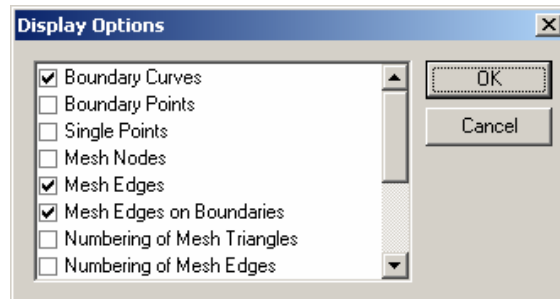
Required Input: Density on right and left hand sides of a fixed point.

Related pages: 1.19-1.21

Related example: Projects 'Grid-1 &2' page 1.20-1.21

*Display is Conditional: Select a fixed point on a boundary in the mesh.

To display click: Boundary Condition Editor (Boundary)/Options/Display Options (☒), or Graphical Display of Results (Graphics)/Options/Display Options (☒)

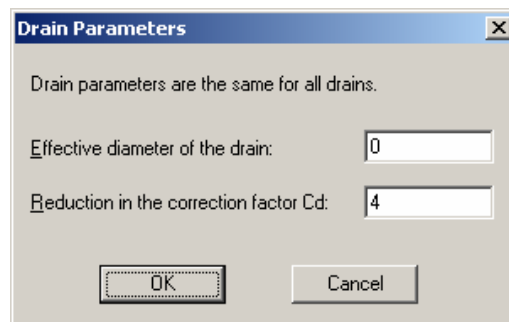


Function: Provides mesh display options.

Required Input: Check the relevant box.

Related pages: 1.46

To display click: Boundary Condition Editor (Boundary) /Condition /Drains/Insert/Left click to select node/right click to end



Function: Assigns drain BC parameters.

Required Input: Effective diameter of the drain and the reduction factor.

Related pages: 1.35

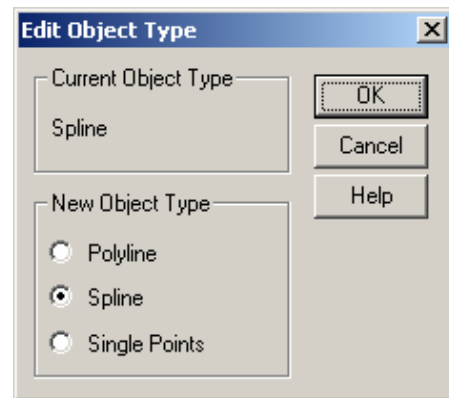
*Display is Conditional: Select 'Rectangular' in 'Geometry Information'.

To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Edit/Geometry/Change Object Type

Function: Changes type of the geometric object.

Required Input: Select a new object type.

Display is Conditional: Select object to be changed.

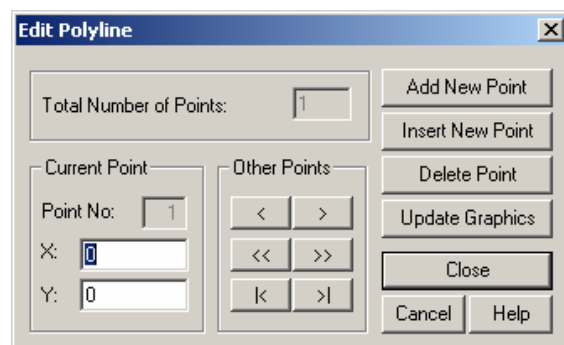


To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Edit/Geometry/Object's Points/Coordinates

Function: Changes polyline points coordinates, adds new or deletes existing points of the polyline numerically.

Required Input: New x and y coordinates of a point.

*Display is Conditional: Select Polyline to be modified.

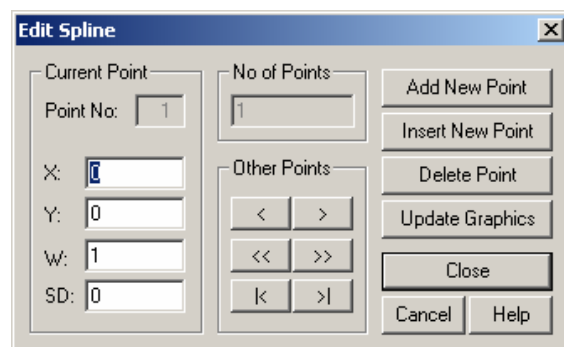


To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Edit/Geometry/Object's Points/Coordinates

Function: Changes spline point coordinates, adds new and delete existing points of the spline numerically.

Required Input: New x and y coordinates of the point, Weight, and Standard Deviation.

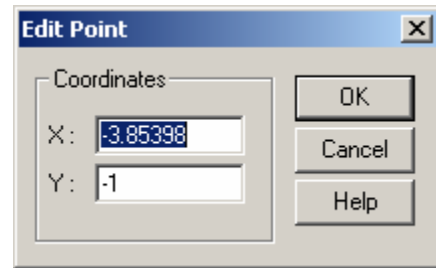
*Display is Conditional: Select Spline to be modified.



To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Edit/Geometry/Object's Point/Move

Function: Moves a point numerically.

Required Input: New x and y coordinates of the point.

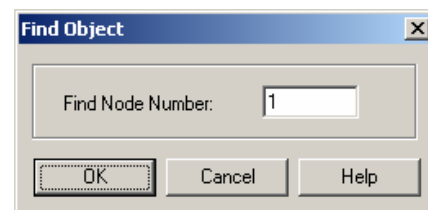
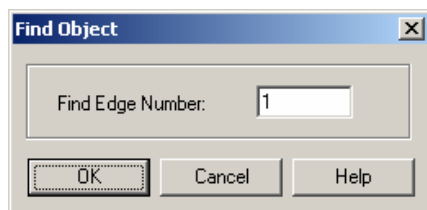
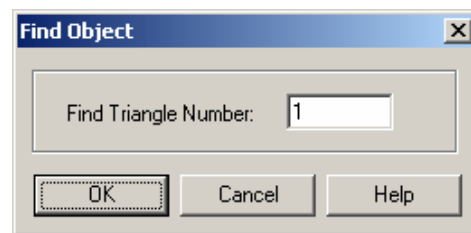


*Display is Conditional: Select 'Tools/Manipulation Mode/Numerical'.

To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Tools/Find, or Boundary Condition Editor (Boundary)/Tools/Find, or Graphical Display of Results (Graphics)/Tools/Find

Function: Locates a triangle, a node, or an edge in the finite element mesh.

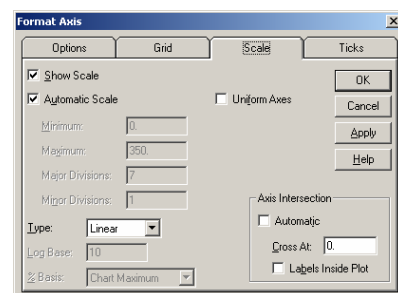
Required Input: Number of a triangle, a node, or an edge of the finite element mesh.



To display click: Right click on a chart/Axis/Axis/Select or double click on an axis in a chart

Function: Formats chart axes.

Required Input: Options/Line type and colour; Grid/Line type, width, and colour; Scale/scale extents and type; Ticks/Location and length.

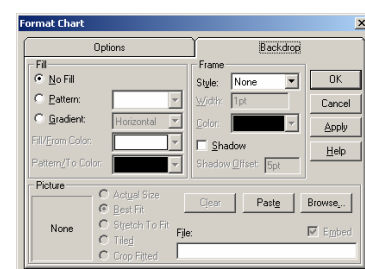


Related pages: 1.51

To display click: Right click on a chart/General or double click on a chart

Function: Formats chart.

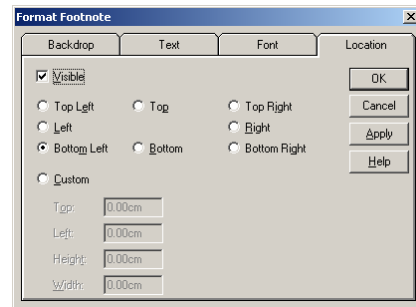
Required Input: Hide/show Title, Legend, Footnote, and 2nd Y-axis; changes fill and frame options.



To display click: Right click on a chart/Footnote

Function: Formats Footnotes.

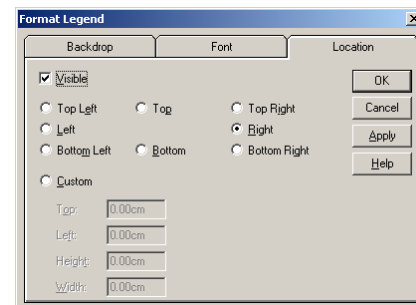
Required Input: Similar to 'Format Legend'.



To display click: Right click on a chart/Legend

Function: Formats Legends.

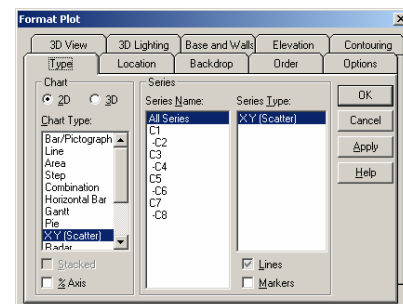
Required Input: Backdrop/set fill and frame options; Font/set font types and size; Location/set the legend location inside the plot and its visibility.



To display click: Right click on a chart/Plot

Function: Formats plots.

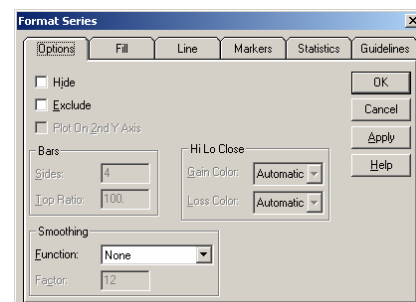
Required Input: Type/Chart type and series; Location/default of specify location; Backdrop sets fills and frames.



To display click: Right click on a chart/Series/Series/Select or double click on a data series in a chart

Function: Formats chart series.

Required Input: Options/Hides or excludes a series/Smooths using two splines options; Line/changes line style, width, and colour; Markers/set style, colour, size, and width; Statistics/plots minimum, maximum, mean, Standard deviation, and regression.

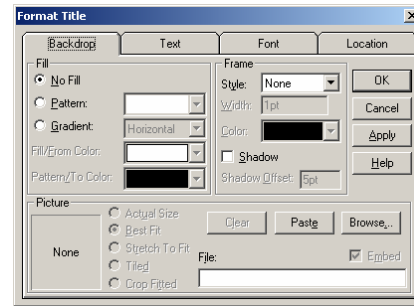


Related pages: 1.49

To display click: Right click on a chart/Title

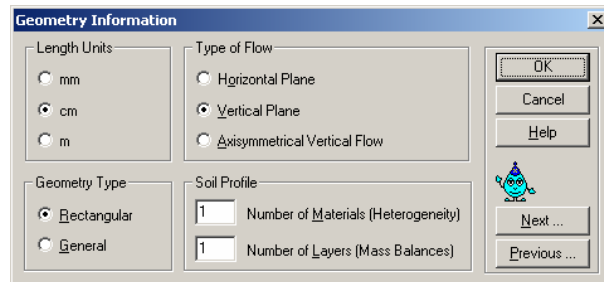
Function: Formats chart Title.

Required Input: Similar option to 'Format Legend'



To display click: Geometry Information (in Pre-Processing)

Function: Selects the length units to be used throughout the project, Type of flow, geometry type (General uses MeshGen-2D package), number of soil types, and number of layers (subregions).



Required Input: Check appropriate boxes, and provide number of materials and layers (subregions).

Related pages: 1.4

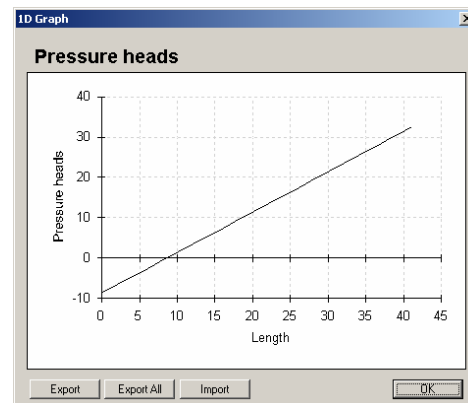
Related example: Project '1D-infil' page x.

To display click: Graphical Display of Results (Graphics)/1D-Graph/Cross Section

Function: Provides a cross section along a boundary or through the domain; displays water content, pressure head, or velocity profiles.

Required Input: Select one of the three quantities you want to display. Click one of the three options to define the boundary.

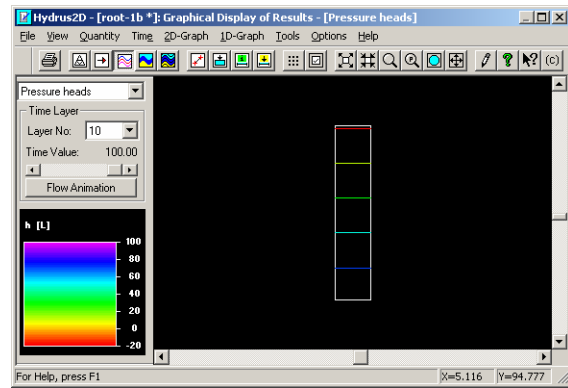
Related pages: 1.45 to 1.48



To display click: Graphical Display of Results (Graphics) (🖨️)

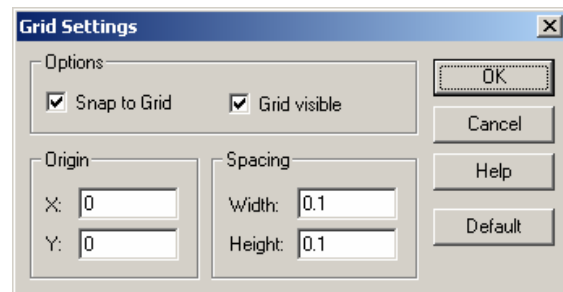
Function: Displays selected variables in a transport domain using spectrum maps, contour maps, velocity vectors, and animations.

Required Input: Select the quantity (water content, velocity, pressure head, concentrations, or temperatures), choose one of the display options, and specify a section.



Related pages: 1.44

To display click: Boundary Condition Editor (Boundary)/Options/Grid Settings, or Graphical Display of Results (Graphics)/Options/Grid Settings (🖨️), or Geometry and FEM Mesh Editor (MeshGen-2D)/Tools/Grid Settings (🖨️)



Function: Turns snap to grid and grid visibility on/off; sets the origin and spacing of the grid.

Required Input: To activate snap and grid visibility, check the appropriate boxes. Enter the x- and y-coordinates of the grid origin, and enter grid spacing.

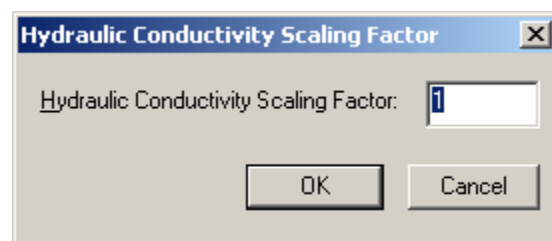
Related pages: 1.48

Related example: Project '1D-infil' page xvi.

To display click: Boundary Condition Editor (Boundary)/Condition/Scaling Factor/Hydraulic Conductivity

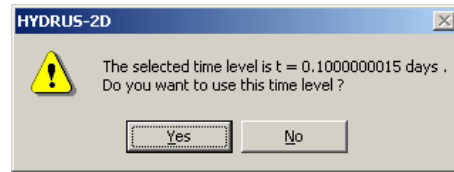
Function: Assigns a scaling factor for the hydraulic conductivity.

Required Input: Value for scaling factor.



Related pages: 1.42; Appendix IV

To display click: Boundary Condition Editor (Boundary)/Condition/Initial Condition/Import Pressure-Water Content



Function: Confirmation for importing initial conditions at a certain Time Level from a previous HYDRUS simulation.

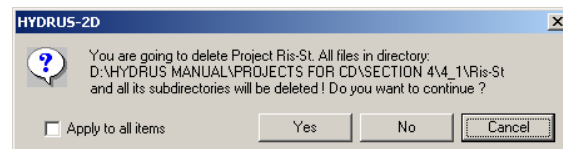
Required Input: Confirms the Time Level at which the initial condition is imported.

Related pages: 1.40; 4.2

Related example: Project 'Dike-St' page 1.40; Project 'Fall-St' page 4.2

*Display is Conditional: Select an 'h.out' file from a previous HYDRUS simulations and specify a Time Level.

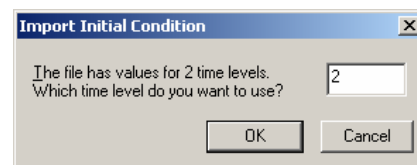
To display click: Project Manager/Projects/Delete



Function: Permanently deletes a project from the current directory.

Related pages: 1.2

To display click: Boundary Condition Editor (Boundary)/Condition/Initial Condition/Import Pressure-Water Content



Function: Imports initial condition at a certain Time Level from a previous HYDRUS simulation.

Required Input: Time Level at which the initial condition is imported.

Related pages: 1.40; 4.2

Related example: Project 'Dike-St' page 1.40; Project 'Fall-St' page 4.2

*Display is Conditional: Select an 'h.out' file from a previous HYDRUS simulations.

To display click: Inverse Solution (In Pre-Processing)

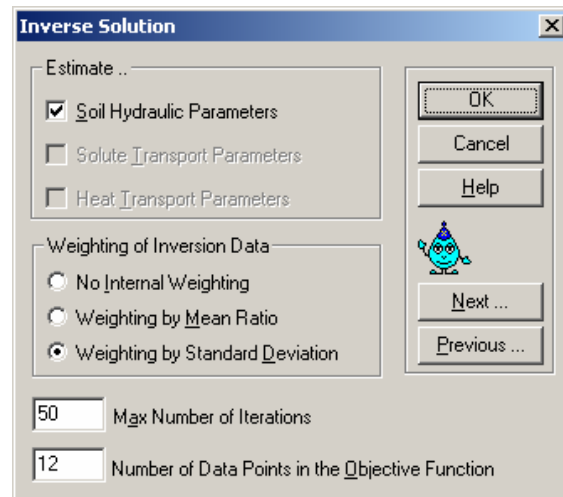
Function: Specifies the parameter types to be optimised and the weighting scheme to be implemented; assigns a limit to the number of iterations and the number of calibration data points.

Required Input: Check the appropriate boxes and enter values as demonstrated in the figure here.

Related pages: 5.1

Related examples: All projects in Section 5.

*Display is Conditional: Check the box 'Inverse Solution' in 'Main Processes'

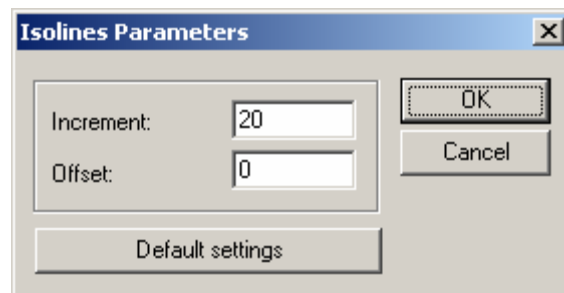


To display click: Graphical Display of Results (Graphics)/Options/Isolines Parameters

Function: Sets isolines parameters.

Required Input: Provide increment value (difference between successive isolines) and offset value (e.g., useful when excluding zero isolines).

Related pages: 1.49



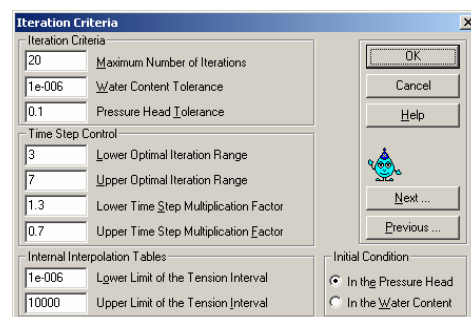
To display click: Water Flow Iteration Criteria (in Pre-Processing)

Function: Sets iteration criteria, time step control, and details of interpolation tables for soil hydraulic parameters; specifies type for initial conditions.

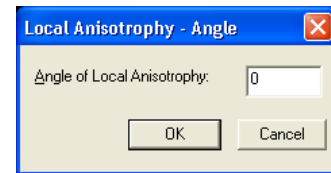
Required Input: Values for all boxes shown in figure here; in most cases these default values need not be changed, especially the 'Time Step Control'. Check the box for 'Initial Condition'.

Related pages: 1.8

Related examples: Project series 'Grad-', page 6.15 to 6.19.



To display click: Boundary Condition Editor
(Boundary)/Condition/Local Anisotropy (↕)/Angle



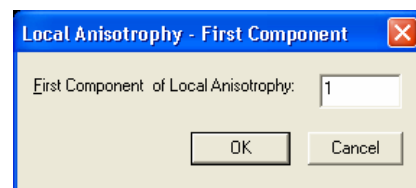
Function: Defines local principal directions along which hydraulic conductivity anisotropy may be defined. The angle is between the principal direction of K_1 and the x-axis of the global coordinate system

Required Input: Angle in degrees, positive counter clockwise.

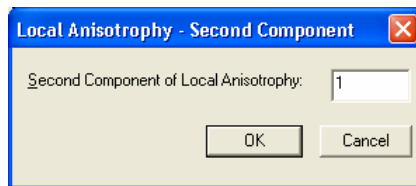
Related pages: I.43

Related example: Project 'Aniso', Example 4.8

To display click: Boundary Condition Editor
(Boundary)/Condition/Local Anisotropy (↕)/First & Second Components



Function: Defines K_1 and K_2 ; they are the principal components of the anisotropy tensor; by default, they coincide with the global coordinate x and z (when angle of local isotropy=0)

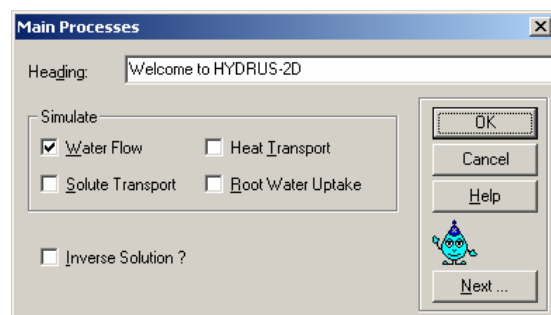


Required Input: values K_1 and K_2 .

Related pages: I.43

Related example: Project 'Aniso', Example 4.8; Project 'DF-Aniso' p. 4.21.

To display click: Main Processes (in Pre-Processing) (👤)



Function: Determines the type of simulation to be conducted. The number of available windows in Pre-Processing and Post-Processing depends on what options are selected here; for example, windows related to root water uptake will not be shown if the option is not checked here.

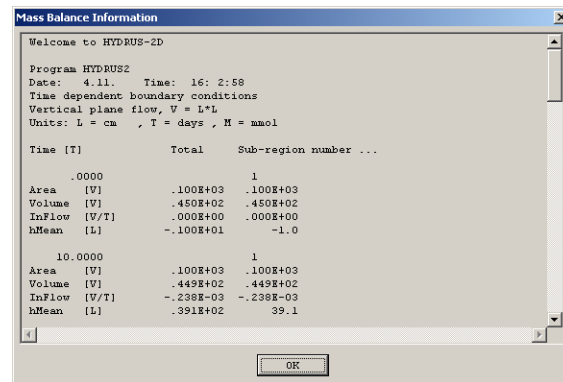
Required Input: Check the box(es) relevant to the simulation. Note that leaving all boxes unchecked prompts HYDRUS to run a steady-state analysis for water flow

Related pages: 1.3

To display click: Maas Balance Information (in Post-Processing)

Function: displays mass balance information at print times for the entire domain and for individual sub-regions (if more than one layer -or subregion- has been specified).

Related pages: 1.52



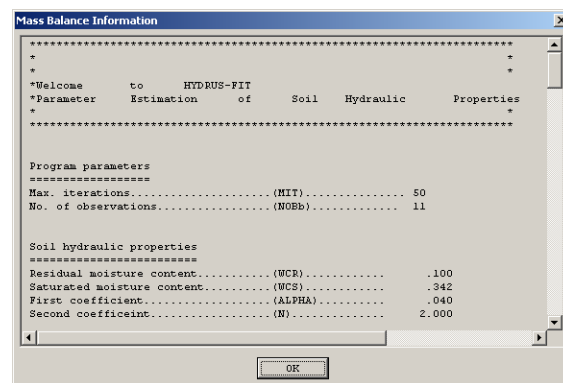
Related examples: Project '1D-Infil' page xviii, Project '2D-Infil' page xx.

To display click: Inverse Solution (in Post-Processing)

Function: Displays inverse solution results.

Related pages: Section 5

*Display is Conditional: Check the box 'Inverse Solution' in 'Main Processes'



To display click: Boundary Condition Editor (Boundary)/Condition/Material Distribution (🇧🇷)

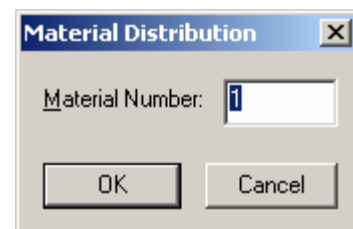
Function: Assigns a material number to a selected group of nodes; only relevant when more than one material is modelled (default value is one).

Required Input: A material number from 1-100 should be specified.

Related pages: 1.35

Related examples: Project '1D-Infil' page xiii.

*Display is Conditional: Select nodes then 'Set Value'.



To display click: Geometry and FEM Mesh Editor (MeshGen-2D) ()

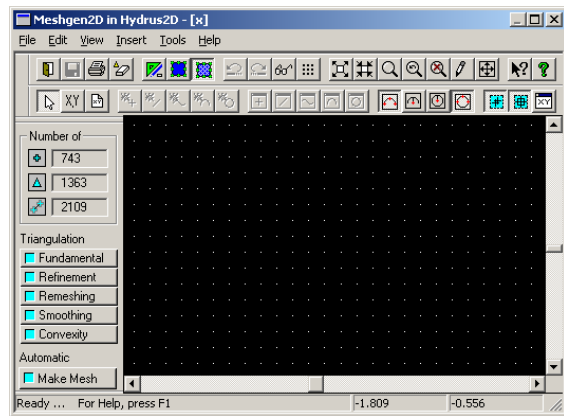
Function: Defines problem domain and discretises it into a finite element mesh.

Required Input: Domain dimensions, number of nodes on the domain boundaries (default=120), and density of mesh.

Related pages: 1.15

Related examples: Project series 'Grid-' pages 1.16 to 1.25

*Display is Conditional: Purchased MeshGen package with HYDRUS-2D; Select 'General' in 'Geometry Information/Geometry Type'.

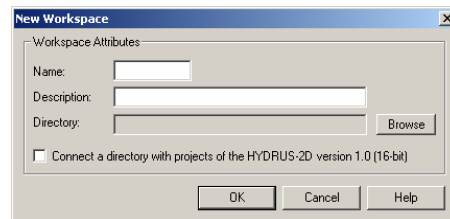
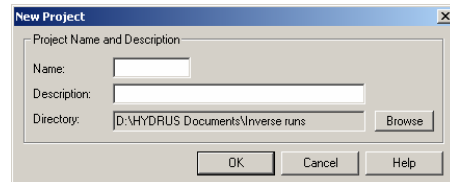


To display click: File/Project Manager/Workspace /New; File/Project Manager/Projects /New

Function: Sets a new working directory (Workspace), or creates new HYDRUS project.

Required Input: Workspace/Project name, description of Workspace/Project, and directory path.

Related pages: 1.1; 1.3



To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/View/Boundary Points/Number of Points (side buttons)

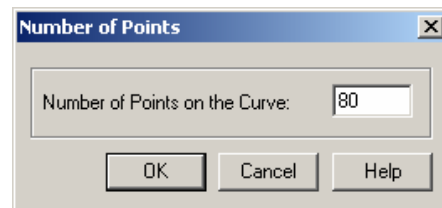
Function: Sets the number of points on an external or internal boundary in the finite element mesh.

Required Input: The total number of points on the selected boundary.

Related pages: 1.19

Related example: Projects 'Grid-1, 2, &3' pages 1.16-1.25

*Display is Conditional: A boundary in the finite element mesh should be selected.



To display click: Observation Points (in Post-Processing)

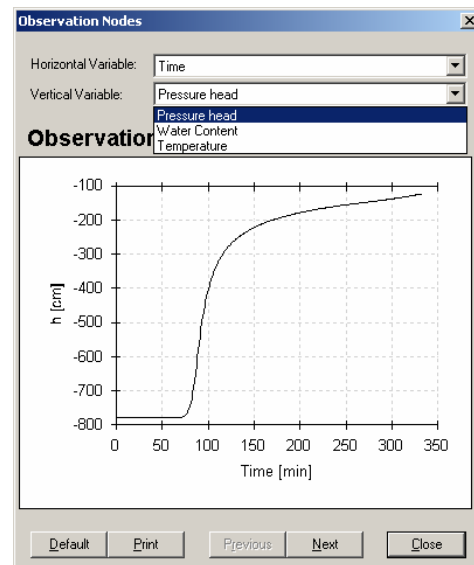
Function: Displays a time-series of pressure heads/water contents/concentrations/temperatures at a specified location defined by 'Observation Points' in the grid. A maximum of 10 observation points is allowed.

Required Input: Select 'Vertical Variable' as shown here.

Related pages: xiv-xv

Related example: Project 'Ris-St' page 4.1 and many others.

*Display is Conditional: Specified 'Observation Points' in 'Boundary Condition Editor/Condition/Observation Node'.

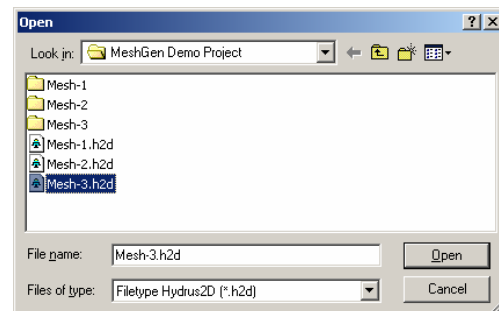


To display click: File/Open (Drop down menu)

Function: Opens existing HYDRUS projects.

Required Input: Select a file with an .h2d extension and click Open.

Related pages: 1.1

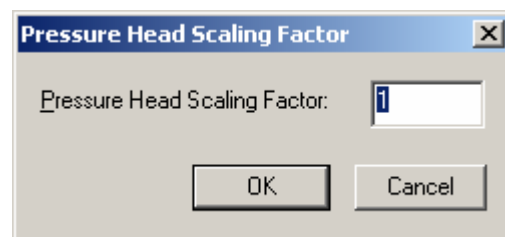


To display click: Boundary Condition Editor (Boundary)/Condition/Scaling Factor/Pressure Head (☰)

Function: Assigns a scaling factor for the Pressure Head.

Required Input: Value for scaling factor.

Related pages: 1.42; Appendix IV



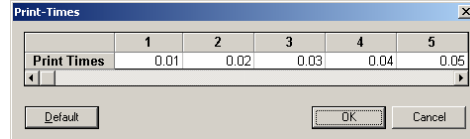
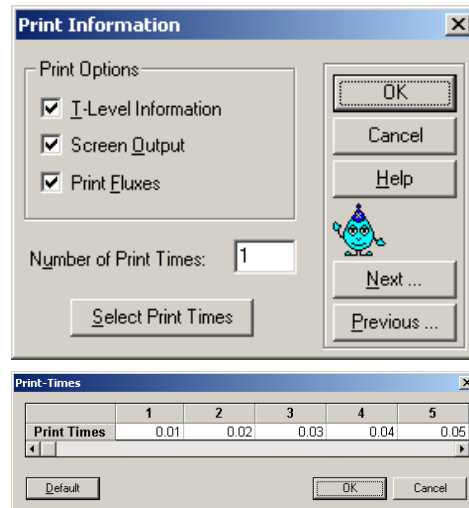
To display click: Print Information (in Pre-Processing)

Function: Determines delivery of output information to output files and screen.

Required Input: Check appropriate box for print options and provide number of print times then select print times (the window below appears).

Related pages: Page 1.6 and 1.7

Related example: Project 'Trial' page 4.26

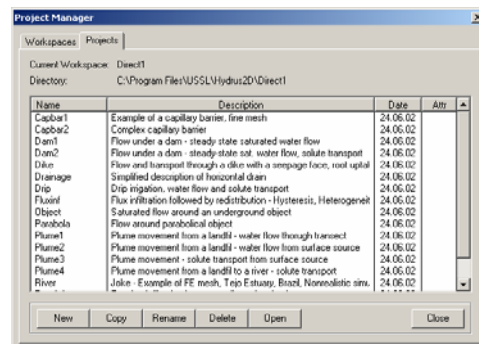


To display click: File/Project Manager/Projects (📁)

Function: Sets new HYDRUS projects, Copies / Renames/ Deletes /Opens existing HYDRUS projects.

Required Input: Click one of the options; New, Copy, Rename, Delete, or Open.

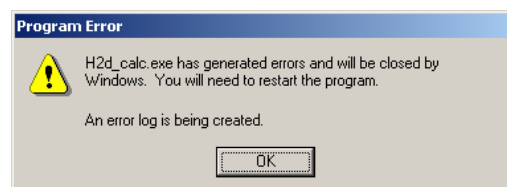
Related pages: 1.2



To display click: Displays automatically after abnormal program termination.

Function: Warns the user of abnormal program termination resulting from a problematic numerical solution.

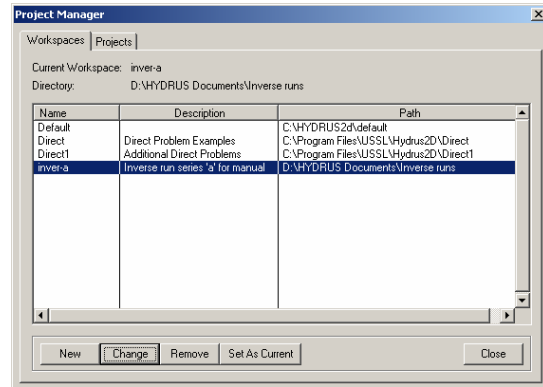
Related pages: 6.8




To display click: File/Project Manager/Workspace ()

Function: Sets a new working directory for HYDRUS projects, changes or removes an existing working directory for HYDRUS projects, and sets a selected workspace as default.

Required Input: Click one of the options; New, Change, Remove, or Set as Default.

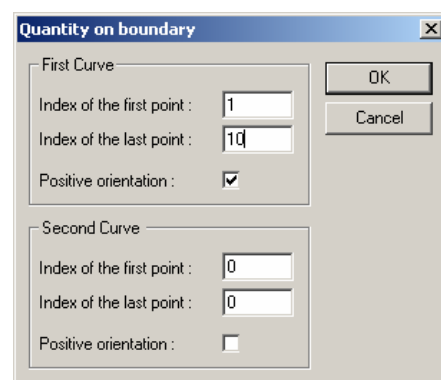


Related pages: 1.1


To display click: Graphical Display of Results (Graphics)/1-D Graph/Boundary Indexes ()

Function: Specifies a range of boundary nodes over which a 1-D Graph is displayed.

Required Input: Starting and ending node numbers representing the boundary section for which the results will be displayed.



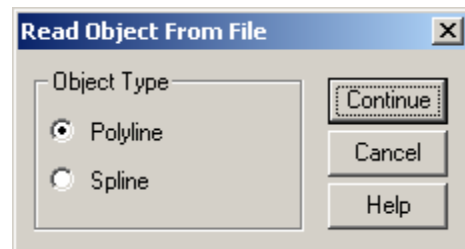
Related page: 1.47

To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert/Object from File ()

Function: Imports coordinates of a set of points representing a spline or a polyline.

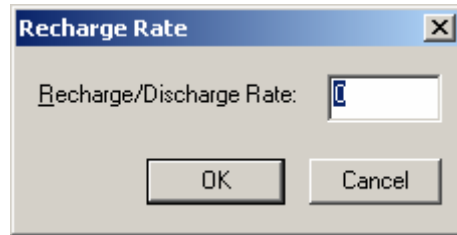
Required Input: Check the appropriate box and have a text file in the suitable format.

Related example: Project 'Grid-3' page 1.22



To display click: Boundary Condition Editor (Graphics)/Condition/Nodal Recharge (✚)

Function: Assigns a recharge or discharge rate for a node anywhere in the domain that acts as a source or sink, respectively.



Required Input: Provide value for recharge (+) or discharge (-) rate.

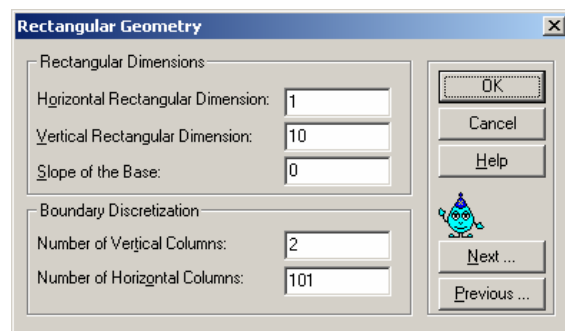
Related pages: 1.43

Related example: Project 'Dewat', Example 4.10 (p. 4.23).

To display click: Geometry and FEM Mesh Editor (🗺) (in Pre-Processing)

Function: Defines problem domain and discretises it into a rectangular finite element mesh.

Required Input: Domain dimensions, slope of the bottom of the domain, number of vertical and horizontal columns for discretizing the domain.



Related pages: 1.13 to 1.15.

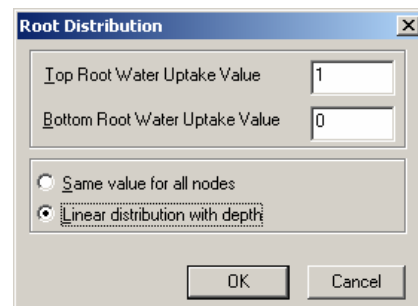
Related examples: Project series '1D-Infil' page xii.

*Display is Conditional: Select 'Rectangular' in 'Geometry Information/Geometry Type'.

To display click: Boundary Condition Editor (Boundary)/Condition/Root Distribution (🌱)

Function: Assigns root density to a set of selected nodes in the domain.

Required Input: Either check 'Same value for all nodes' and provide one density value, or, check 'Linear distribution with depth' and provide two density values.



Related pages: 3.2.

Related examples: Project 'Root-4' page 3.6.

*Display is Conditional: Check 'Root Water Uptake' in 'Main 'Processes'', and, Select nodes then 'Set Value'.

To display click: Boundary Condition Editor (Boundary)/Options/ Parameters for Root Distribution

Function: Assigns 1-D or 2-D root distribution parameters.

Required Input: Parameters as shown in figure here.

Related pages: III.4-III.6

*Display is Conditional: Check 'Root Water Uptake' in 'Main 'Processes'.

To display click: Root Water Uptake Models (in Pre-Processing)

Function: Selects the water uptake reduction model.

Required Input: Check appropriate box

Related pages: Page 3.2, and Appendix III.

Related examples: See Section 3.

*Display is Conditional: Check 'Root Water Uptake' in 'Main 'Processes'

To display click: Root Water Uptake Models - Pressure Head Reduction (in Pre-Processing)

Function: Sets pressure heads that control root water uptake for Feddes' model.

Required Input: Values shown in figure (database provides suggested values).

Related pages: Page 3.2, and Appendix III.

Related examples: See Section 3.

*Display is Conditional: Check 'Root Water Uptake' in 'Main 'Processes', and checked 'Feddes' in 'Root Water Uptake Model/Root Water Uptake Model'

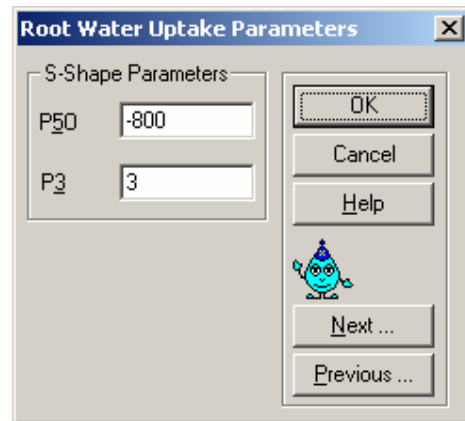
To display click: Root Water Uptake Models - Pressure Head Reduction (in Pre-Processing)

Function: Sets reduction parameters for an S-Shaped model.

Required Input: P50 and P3.

Related pages: Page 3.3, and Appendix III.

*Display is Conditional: Check 'Root Water Uptake' in 'Main Processes', and checked 'S-Shape' in 'Root Water Uptake Model/Root Water Uptake Model'

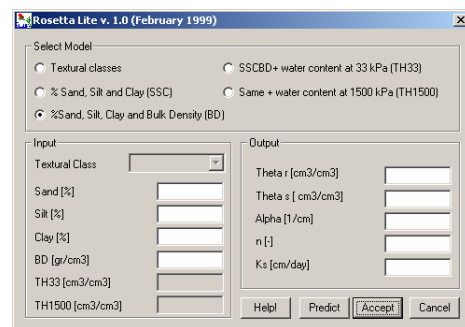


To display click: Water Flow – Soil Hydraulic Parameters (in Pre-Processing)/Neural Network Prediction

Function: Provides predictions for soil hydraulic parameters from pedotransfer functions.

Required Input: The input depends on the selected model; soil textural class, or sand/silt/clay contents with or without bulk density, and the latter with 1 or 2 points on the water retention curve.

Related pages: Page 1.11



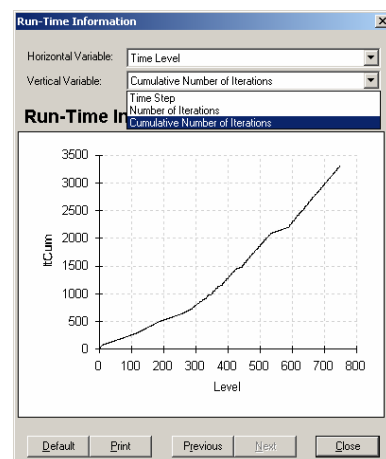
To display click: Run Time Information (in Post-Processing)

Function: Provides graphical display of information related to the numerical solution such as number of iterations.

Required Input: Choice of vertical variable: time step, number of iterations, or cumulative number of iterations; choice of horizontal variable: time or time level.

Related pages: 1.52

Related examples: Project 'Mesh-1 & 2', page 6.1.

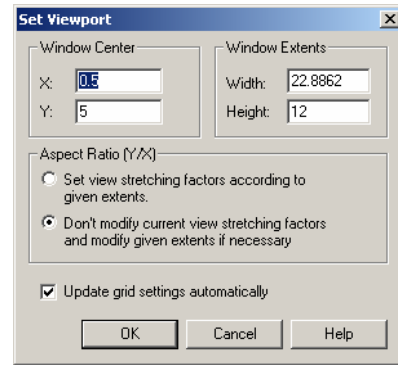


To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/View/Viewport (+), or Boundary Condition Editor (Boundary)/View/Viewport, or Graphical Display of Results (Graphics)/View/Viewport

Function: Set the screen limits and centre; also sets aspect ratio options

Required Input: Coordinates for window centre and extents, check appropriate option for stretching.

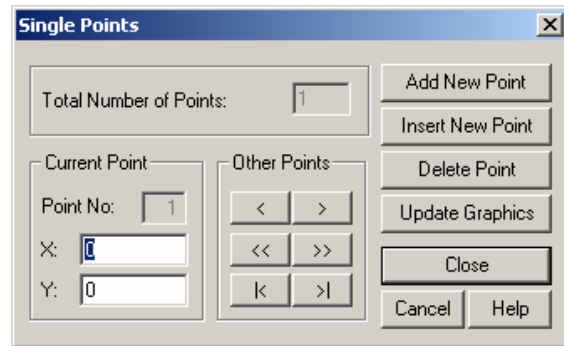
Related pages: 1.15



To display click: Geometry and FEM Mesh Editor (MeshGen-2D)/Insert/Single Point/Numerically (+)

Function: Adds a point to boundary.

Required Input: x and y coordinates of a point.



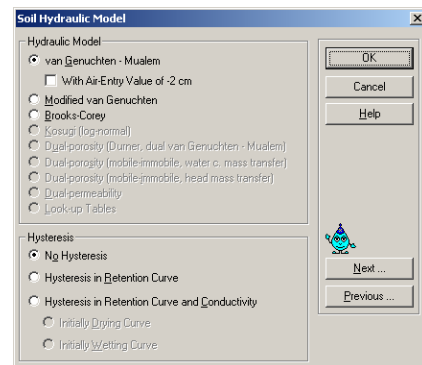
To display click: Soil Hydraulic Model (in Pre-Processing)

Function: Sets one of three soil hydraulic models; hysteresis option available only with van Genuchten-Mualem model.

Required Input: Check appropriate box for soil hydraulic model.

Related pages: 1.10; Appendix I.

Related examples: All Project examples; hysteresis example page 4.8.

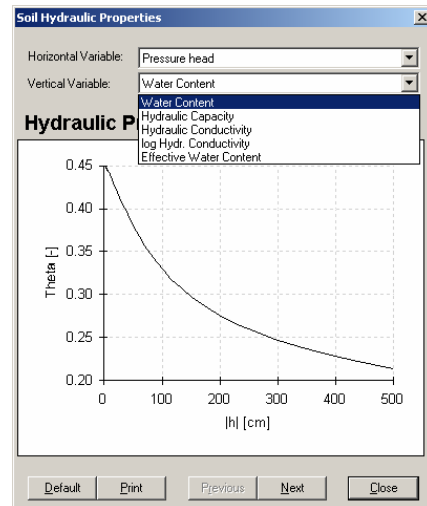


To display click: Soil Hydraulic Properties (in Post-Processing)

Function: Displays soil water retention and hydraulic conductivity function.

Required Input: Select required 'Vertical and Horizontal Variable'.

Related pages: 1.51

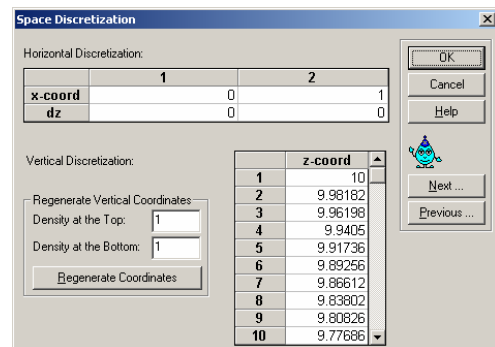


To display click: Geometry and FEM Mesh Editor (in Pre-Processing)

Function: Changes default mesh discretization.

Required Input: May manually change coordinates; enter density values to alter mesh density at top or bottom.

Related pages: 1.14



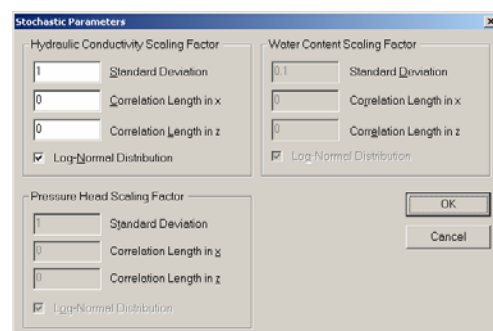
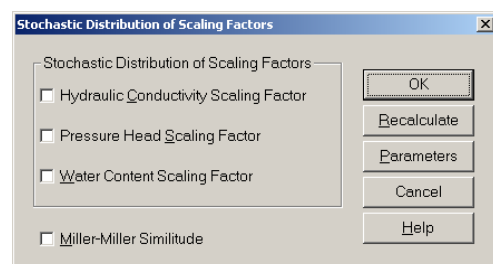
*Display is Conditional: Select 'Rectangular' in 'Geometry Information/Geometry Type'.

To display click: Boundary Condition Editor (Boundary)/Options/Stochastic Distribution of S.F.

Function: Assigns stochastic distribution to scaling factors.

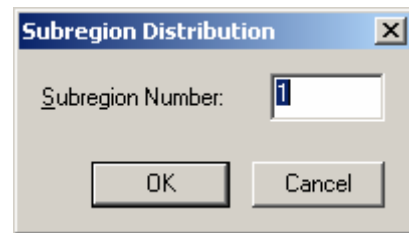
Required Input: Check appropriate boxes and click 'Parameters', the second window here will appear; enter values for standard deviation, and correlation length in x and z directions.

Related pages: 1.42; Appendix IV



To display click: Boundary Condition Editor (Boundary)/Condition/Subregions (📄)

Function: Assigns a subregion number to a selected group of elements; only relevant when more than one subregion is modelled (default value is one).



Required Input: A subregion number from 2-10 should be specified.

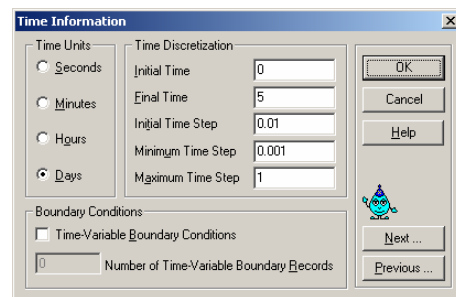
Related pages: 1.41

Related examples: Project '1D-Infil' page xiv

*Display is Conditional: Select nodes then 'Set Value'.

To display click: Time Information (in Pre-Processing)

Function: Defines time units, time discretization, and number of time-variable boundary conditions (if any).



Required Input: Check appropriate box for time units, and enter time discretization details; if time-variable boundary conditions are checked then provide a number.

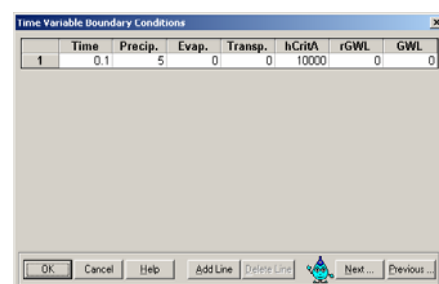
Related pages: 1.5

Related examples: Project in Section 6 show the significance of 'Time Discretization'

To display click: Variable Boundary Conditions (in Pre-Processing)

Function: Defines time-variable BC such as precipitation (rainfall), evaporation, transpiration, and time-variable pressure head or flux.

Required Input: Time series for relevant boundary condition.



Related pages: 1.12; 1.29-1.30; 1.33-1.34

Related examples: Project 'Ris-St' page 4.1; Project 'Evap-1' page 4.3; and Project 'Tension Disc Infiltrimeter' page 4.21

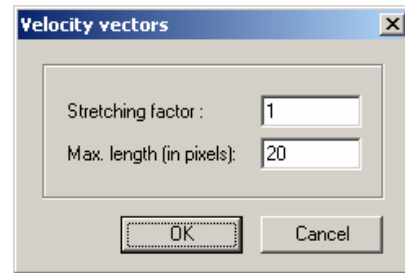
*Display is Conditional: Check 'Variable Boundary Condition' in 'Time Information'.

To display click: Graphical Display of Results (Graphics)/Options/Velocity Vectors Parameters

Function: Scales arrows representing velocity vectors.

Required Input: As shown in figure here.

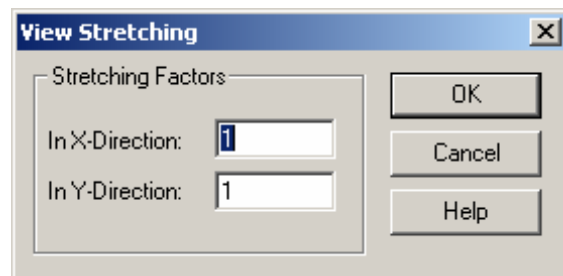
*Display is Conditional: Check '2D-Graphs/Velocity Vectors'.



To display click: Boundary Condition Editor (Boundary)/View/Stretching, or Graphical Display of Results (Graphics)/View/Stretching, or Geometry and FEM Mesh Editor (MeshGen-2D)/View/Stretching

Function: Stretches the domain. The option is useful when the aspect ratios of the domain and the screen are significantly different.

Required Input: Stretching factors in x- or y-directions; note that when the factor is < 1 the mesh is stretched in that direction.

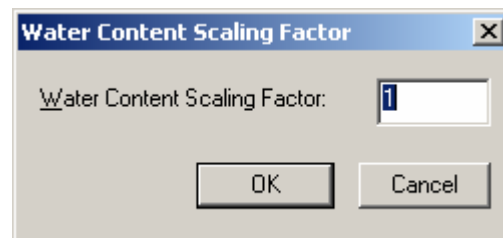



To display click: Boundary Condition Editor (Boundary)/Condition/Scaling Factor/Water Content

Function: Assigns a scaling factor for the water content.

Required Input: Value for scaling factor.

Related pages: 1.42; Appendix IV

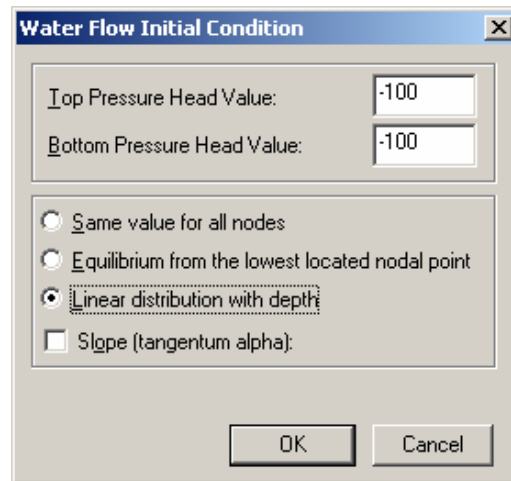


To display click: Boundary Condition Editor (Boundary)/Condition/Initial Condition/Pressure Head-Water Content ()

Function: Assigns initial pressure heads or water contents.

Required Input: One value if 'Same for all nodes' is checked; Top and Bottom values if one of the other options is checked; Specify slope of the initial condition.

Related pages: 1.37-1.38



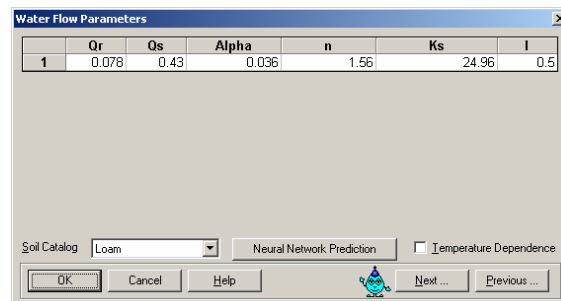
To display click: Water Flow - Soil Hydraulic Parameters (in Pre-Processing)

Function: Defines the soil water retention parameters, the saturated hydraulic conductivity, and L (turtuosity parameter).

Required Input: Water retention parameters and measured saturated hydraulic conductivity; required values vary according to the selected hydraulic model.

Related pages: 1.11; Appendix I.

Related examples: All example Projects



To display click: Water Flow - Soil Hydraulic Parameters (in Pre-Processing)

	Qr	Qs	Alpha	n	Ks	l
Initial Estimate	0.1	0.3418	0.04	2	0.0051	0.5
Minimum Value	0.05	0.3	0.005	1	0.001	0
Maximum Value	0.15	0.4	0.2	2.5	0.015	0
Fitted ?	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Soil Catalog: Neural Network Prediction Temperature Dependence

OK Cancel Help Next ... Previous ...

Function: Defines soil water retention parameters, the saturated hydraulic conductivity, and L (tortuosity parameter); if 'Fitted' is checked, those values are used as initial values for the inverse solution; sets upper and lower limits for the optimised parameters. Determines which parameters are optimised.

Required Input: Water retention parameters, the saturated hydraulic conductivity, and the tortuosity parameter; minimum and maximum limits for the optimised parameters; along the line 'Fitted', check the parameters to be optimised.

Related pages: Page 5.2.
















Related examples: All examples in Section 5.

*Display is Conditional: Check the box 'Inverse Solution' in 'Main Processes'.

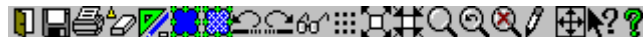
APPENDIX VIII TOOLBARS





















Main Program Toolbars:



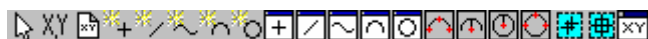
	New Project (File Menu)
	Open a Project (File Menu)
	Project Manager (File Menu)
	Save Project (File Menu)
	Main Processes (Pre-Processing Menu)
	Geometry and Finite Element Mesh Editor (Pre-Processing Menu)
	Boundary and Initial Conditions (Pre-Processing Menu)
	Run HYDRUS (Calculation Menu)
	Graphical Display of Results (Post-Processing Menu)
	Cascade Window (Window Menu)
	Tile Windows Horizontally (Window Menu)
	Tile Windows Vertically (Window Menu)
	HYDRUS Help Topics (Help Menu)
	Context Sensitive Help (Help Menu)
	License Details (Help Menu)

MeshGen2D Toolbars 1:



	Close MeshGen2D (File Menu)
	Save Mesh (File Menu)
	Print (File Menu)
	Delete Mesh (Edit Menu)
	Edit Geometry (Edit Menu)
	Edit Boundary Points (Edit Menu)
	Edit FEM Mesh (Edit Menu)
	Undo (Edit Menu)
	Redo (Edit Menu)
	Consistency Check (Tools Menu)
	Grid Settings (Tools Menu)
	Enlarge View (View Menu)
	Reduce View (View Menu)
	Zoom View (View Menu)
	Previous View (View Menu)
	View All (View Menu)
	Redraw (View Menu)
	Set Viewport (View Menu)
	Context Sensitive Help (Help Menu)
	MeshGen2D Help Topics (Help Menu)

MeshGen2D Toolbars 2:







	Graphical Manipulation Mode (Tools Menu)
	Numerical Manipulation Mode (Tools Menu)
	Read Object from File (Insert Menu)
	New Points Graphically (Insert Menu)
	New Line Graphically (Insert Menu)
	New Spline Graphically (Insert Menu)
	New Arc Graphically (Insert Menu)
	New Circle Graphically (Insert Menu)
	New Points Numerically (Insert Menu)
	New Line Numerically (Insert Menu)
	New Spline Numerically (Insert Menu)
	New Arc Numerically (Insert Menu)
	New Circle Numerically (Insert Menu)
	Arc by Three Points (Insert Menu)
	Arc by Center, Radius, and Two Angles (Insert Menu)
	Circle by Center and Radius (Insert Menu)
	Circle by Three Points (Insert Menu)
	Select Duplicated Points (Insert Menu)
	Pick Existing Points (Tools Menu)
	Edit Objects Coordinates (Tools Menu)

Boundary Conditions Editor Toolbars:
























	Close BC Editor (File Menu)
	Print (File Menu)
	Save (File Menu)
	Display Options (Options Menu)
	Water Flow BC (Condition Menu)
	Solute Transport BC (Condition Menu)
	Heat Transport BC (Condition Menu)
	Material Distribution (Condition Menu)
	Root Uptake Distribution (Condition Menu)
	Nodal Recharge (Condition Menu)
	Scaling Factors (Condition Menu)
	Initial Conditions (Condition Menu)
	Anisotropy (Condition Menu)
	Subregions (Condition Menu)
	Observation Nodes (Condition Menu)
	Nodal Drains (Condition Menu)
	Flowing Particle (Condition Menu, currently inactive)
	Zoom In (View Menu)
	Zoom Out (View Menu)
	Zoom View (View Menu)
	Previous View (View Menu)
	View All (View Menu)

-  Redraw (View Menu)
-  HYDRUS Help Topics (Help Menu)
-  Context Sensitive Help (Help Menu)
-  License Details (Help Menu)

Graphical Display of Results Toolbars:



-  Print (File Menu)
-  Display FE Mesh (2-D Graph Menu)
-  Velocity Vectors (2-D Graph Menu)
-  Isolines (2-D Graph Menu)
-  Spectral Map (2-D Graph Menu)
-  Spectral Map and Isolines (2-D Graph Menu)
-  1-D Graph along Cross-Section (1-D Graph Menu/Cross Section)
-  1-D Graph along Whole Boundary curve (1-D Graph Menu/Boundary Whole Curve)
-  1-D Graph along a Boundary Between Two Given Points (1-D Graph Menu/Boundary Indexes)
-  1-D Graph along a Boundary Between Two Graphically Selected Points (1-D Graph Menu/Boundary Indexes)
-  Grid Settings (Options Menu)
-  Display Options (2-D Graph Menu)
-  Zoom In (View Menu)
-  Zoom Out (View Menu)
-  Zoom View (View Menu)
-  Previous View (View Menu)
-  View All (View Menu)
-  Redraw (View Menu)
-  HYDRUS Help Topics (Help Menu)
-  Context Sensitive Help (Help Menu)
-  License Details (Help Menu)

ALPHABETICAL INDEX

Description	Page for Examples	Page in Appen. VII	Page in various Sections
1D-Graph			1.45-1.48
1D-Graph/boundary indexes			1.47
1D-Graph/boundary selection			1.46
1D-Graph/Boundary whole curve			1.45
1D-Graph/Cross section			1.48
Abnormal HYDRUS termination	6.8	VII.20	
Actual root water uptake	3.4/3.8		3.2
Air entry value (AEV)			I.3/I.5/I.10
Alpha-water flow parameter			1.11/I.3/ I.10
AlphaW-wetting Alpha parameter	4.8		
Anisotropy	4.18	VII.16	1.43
Arc	1.17	VII.1	
Aspect ratio			1.16
Axisymmetric flow	4.21-4.24		1.4
Boundary conditions (see water flow Boundary conditions)			1.26
Brooks and Corey model			1.10/I.5/ I.8
Calibration data (data points in objective function)	V.8		5.1-5.3
Calibration data/choice of data			V.8 & 9
Capillary barrier	4.6/4.21		
Change workspace			1.2
Check Geometry			1.25
Circle	1.17/1.24	VII.3	
Concentration	VI.2/VI.6	VII.5	
Confidence limit			V.4
Consistency check			1.25
Constraints on hysteresis model (inverse mode solution)			5.2
Convert to ASCII			1.53/2.5
Copy project	1.40/4.2		1.2
Correlation length/ Stochastic parameters			IV.5
Correlation matrix/Inverse solution			V.6
Cumulative water boundary fluxes/Graphical display of results			1.50
Curve type	1.24	VII.6	
Data for inverse solution			5.2/5.3
Density at the bottom			1.15
Density at the top			1.15
Density of boundary points/MeshGen	1.19-1.21	VII.8	
Disc infiltrometer	4.21		1.27/1.28
Dispersivity	VI.2		
Drain BC		VII.8	1.35
Drying water flow parameters/hysteresis	4.8		
Evaporation	4.3		1.13 Appen.-II
Evaporation - advance of drying boundary	4.3/4.4		
Evaporation - effect of hCritA	II.3		
Evaporation - effect of mesh density	4.4		
Evaporation - effect of relative humidity			II.2
Evaporation - effect of suction			II.1

Description	Page for Examples	Page in Appen. VII	Page in various Sections
Evaporation - shape of drying curve			II.1
Evaporation - vapour flow			II.2
Feddes root water uptake reduction model			3.2/III.1
Final time			1.5
General geometry	1.15		1.4
Geometry Information window		VII.12	1.4
Geometry Type/Geometry Information			1.4
Goodness of fit			V.5
Graph-1D			1.45-1.48
Graphical display of results window		VII.13	1.44
Grid settings window		VII.13	1.48
Guidelines on inverse modelling			5.24/5.25
GWL	4.1/4.21		1.13/1.29
hCritA-Evaporation parameter	II.3		1.13
Heading/main processes	xvi		1.3
Heterogeneous profile	x/xiv		1.5/1.35
h_k parameter for Vogel & Cislerova model			1.7
Horizontal flow	4.25		1.5
HYDRUS (Multiple simulations)	4.27		
Hysteresis	4.8		1.10
Import initial condition from previous HYDRUS simulation	1.40/4.2/VI.7		
In the pressure head/Iteration Criteria		VII.15	1.8/1.10
In the water content /Iteration Criteria	xii	VII.15	1.8/1.10
Initial condition/Iteration Criteria			1.8/1.10
Initial conditions			1.37/1.40
Initial parameters for inverse solution	5.15/5.19		5.1
Initial time	4.2		1.5
Initial time step	6.9		1.5
Insert Arc/MeshGen	1.17	VII.1	
Insert circle numerically/MeshGen	1.17/1.24		
Insert fixed point/MeshGen	1.21		
Insert line numerically/MeshGen	1.17		
Insert object from file/MeshGen	1.22	VII.21	
Insert periodical condition			1.25
Insert Spline/MeshGen	1.23		
Internal interpolation tables/ Iteration Criteria	6.14		1.10
Inverse solution window		VII.15	5.1
Inverse solution/in Main processes			1.3/5.1
Isolines parameters		VII.15	1.49
Iteration Criteria window		VII.15	1.8
Key predictions in inverse simulations	5.18		
K_k parameter for Vogel & Cislerova model			1.7
K_s - water flow parameter (saturated hydraulic conductivity)			1.11/I.10
K_sW -wetting saturated hydraulic conductivity	4.9		
L-water flow parameter (Totuosity parameter)			1.11-5.14
Layers (or subregions)	xv		1.41
Length units/ Geometry Information			1.4
Line	1.17		
Lower-limit of the tension interval	4.9/6.14		1.10
Main processes window		VII.16	1.3
Mass balance information/Graphical display of results	6.6/xix		1.52
Material distribution	xiv		1.35/1.36
Maximum number of iterations	6.8		1.8/1.9

Description	Page for Examples	Page in Appen. VII	Page in various Sections
Maximum number of iterations/inverse solution			5.1
Maximum time step			1.5/1.9
Merge object/MeshGen			1.25
Mesh density	4.4/4.16		1.15/1.20
MeshGen	1.15-1.25		1.15
Minimum time step	6.3		1.5/1.9
Modified van Genuchten model (Vogel & Cislserova)			1.10/1.5/1.7
m-parameter (van Genuchten model)			1.2/1.4
Multiple HYDRUS simulations	4.27		
Neural Network Prediction (Rosetta)			1.11
New project			1.2
New workspace			1.2
Nodal drain			1.35
Nodal recharge	4.23		1.43
Non-uniqueness of inverse solution	Sec. 5.2.1		5.4/V.6
Normalised water content			I.3
Number of boundary points/MeshGen	1.19	VII.18	
Number of data points in objective function			5.1
Number of layers	x/xv		1.4/1.41
Number of materials	x/xiv		1.5/1.35
Number of print times	4.26		1.7
n-water flow parameter			1.11/I.3/ I.10
Objective function/ Inverse solution			V.1
Observation nodes	xv		1.43
Open project window		VII.19	1.1
Output files			2.1
o A_level.out	2.11		
o Balance.out	2.11		2.4
o Boundary.out	2.8/2.12		2.1
o Check.out			2.3
o Cum_Q.out	2.11		2.3
o Fit.out			2.5
o H.out (Optionally converted to an ASCII file h.txt)			2.5
o h_mean.out			2.2
o MeshTria.000 (Optionally convert to an ASCII file MeshTria.txt)			2.5
o ObsNod.out	2.7		2.2
o Run_Inf.out			2.4
o Th.out (Optionally converted to an ASCII file Th.txt)			2.5
o V.out (Optionally converted to an ASCII file v.txt)			2.5
o V_mean.out	2.7/2.10/2.15		2.2
Parameter interaction in inverse solution	5.18		
Pick existing points			1.25
Pointer to vector of solute BC	VI.3		
Precipitation			1.13
Pressure head tolerance	4.9/4.16		1.8
Pressure heads/Graphical display of results			1.50
Print fluxes			1.6
Print information window		VII.20	1.6
Print Options/Print Information			1.6
Program error/abnormal HYDRUS termination	6.8	VII.20	

Description	Page for Examples	Page in Appen. VII	Page in various Sections
Project manager window		VII.20	1.1
Pulse duration	VI.2		
Q_m -(Theta_m) parameter Vogel & Cislrova model (Modified van Genuchten)			1.5
Q_r -water flow parameter (residual water content)			1.11/I.2&3
Q_s - water flow parameter (saturated water content)			1.11/I.3
$Q_s W$ -wetting saturated water content	4.8		
Rectangular Geometry	1.13		1.4
Relative humidity			II.2
Remove workspace			1.2
Rename project			1.2
Residual water content (see Q_r ; Theta_r)			1.11/I.2&3
RGWL			1.13/1.30
Root density			III.3
Root distribution parameters			III.4 to 6
Root distribution window	3.6	VII.22	3.2
Root water uptake model window		VII.23	3.2
Root water uptake parameters		VII.23	3.2/III.1
Rosetta window		VII.24	1.11
Run time information/Graphical display of results	6.2-6.4		1.52
Runoff (see Surface flux higher than K_s)	4.17		1.34
Saturated water content (Q_s ; Theta_s)			1.11/I.3
Scaling factors			1.42/IV.1
Screen capture			1.45
Screen output			1.6
Select by rhomboid			1.36
Select print times			1.7
Sensitivity analysis for water retention parameters			I.9 to I.12
Set viewport window	1.17/1.20	VII.25	1.15
Set workspace as current			1.2
Similar media			IV.1
Simulate/main processes			1.3
Soil hydraulic model window		VII.25	1.10
Soil hydraulic properties /Graphical display of results			1.51
Soil Profile/Geometry Information			1.4
Solute transport BC	VI.2		
Solute transport parameters	VI.2		
Spline	1.22 & 23		
Split object/MeshGen			1.25
S-shaped parameters- Root water uptake			3.3/ III.2
Standard deviation/ Stochastic parameters			IV.4
Steady state analysis	1.39/1.40/4.15		1.3/1.39
Stochastic distribution of scaling factor window		VII.26	1.42/IV.3
Stochastic parameters window		VII.26	1.42/IV.4
Subregions (or layers)	x/xv		1.4/1.41
Surface flux higher than K_s	4.17		1.34
Tension disc infiltrometer	4.21		1.27/1.28
Theta_m (Q_m) parameter Vogel & Cislrova model (Modified van Genuchten)			1.5
Theta_r water flow parameter (residual water content Q_r)			1.11/I.2/I.3
Theta_s water flow parameter (saturated water content Q_s)			1.11/I.3
Time discretization			1.5/1.9
Time information window		VII.27	1.5
Time Step Control/Iteration Criteria			1.8/1.9

Description	Page for Examples	Page in Appen. VII	Page in various Sections
Time units			1.5
Time-variable BC, number of records allowed			1.34
Time-variable boundary condition checkbox/Time information			1.5
Time-variable boundary condition/window		VII.27	1.12/1.29/ 1.30/1.33/ 1.34/
Tipping bucket rainfall data			1.33/1.34
T-level information			1.6/2.1
Tools/find		VII.10	1.25
Tools/point picking			1.25
Tortuosity parameter			1.11-5.14
Transpiration	Section 3		1.13/Sec-3
Trouble shooting for HYDRUS-2D			Section 6
o Effect of minimum time step	6.1-6.3		
o Effect of abrupt intense infiltration	6.4-6.7		
o Unstable initial conditions/ Boundary + saturation effects	6.8-6.9		
o Infiltration into clay/ Model + flux/ K_s effects	6.10-6.11		
o Infiltration into clay/ Time discretization + Iteration criteria effects	6.12-6.14		
o Effect of high initial press. head gradient	6.15-6.19		
Type of flow/ Geometry Information	4.21/4.25		1.4
Upper-limit of the tension interval	4.9/6.14		1.10
Validation of inverse results	5.8/5.9/5.13		5.8
van Genuchten model			1.10/I.2/I.6
van Genuchten model with Air-entry value of -2 cm			1.10/I.4/I.7
Vapour flux during evaporation			II.3
Velocity vectors	4.19		
Vertical flow	4.1 to 4.20		1.4
Vogel & Cislerova model (Modified van Genuchten)			1.5/I.7
Water boundary fluxes/Graphical display of results			1.50
Water content tolerance	3.5/4.9/4.16/6.13		1.8
Water flow Boundary conditions			1.27
o No Flux			1.27
o Constant head	4.12	VII.5	1.27/1.28
o Constant flux	5.4	VII.4	1.29
o Variable pressure	4.1/4.21		1.29
o Variable flux			1.30
o Free drainage	4.6		1.31
o Deep drainage		VII.7	1.31
o Seepage face	4.12/4.18		1.31/1.32
o Atmospheric	5.17/4.18		1.33
o Drain			1.35
Water flow parameters inverse solution window		VII.30	5.2
Water flow parameters window		VII.29	1.11
Water flow-initial conditions		VII.29	1.38
Water retention curve			I.1 to I.6
Weighted residual			V.2
Weighting of inversion data/Inverse solution	V.7/V.8		5.1
Weighting parameters			V.1/V.2/ V.7
Wetting water flow parameters/hysteresis	4.8		
Workspace			1.1

