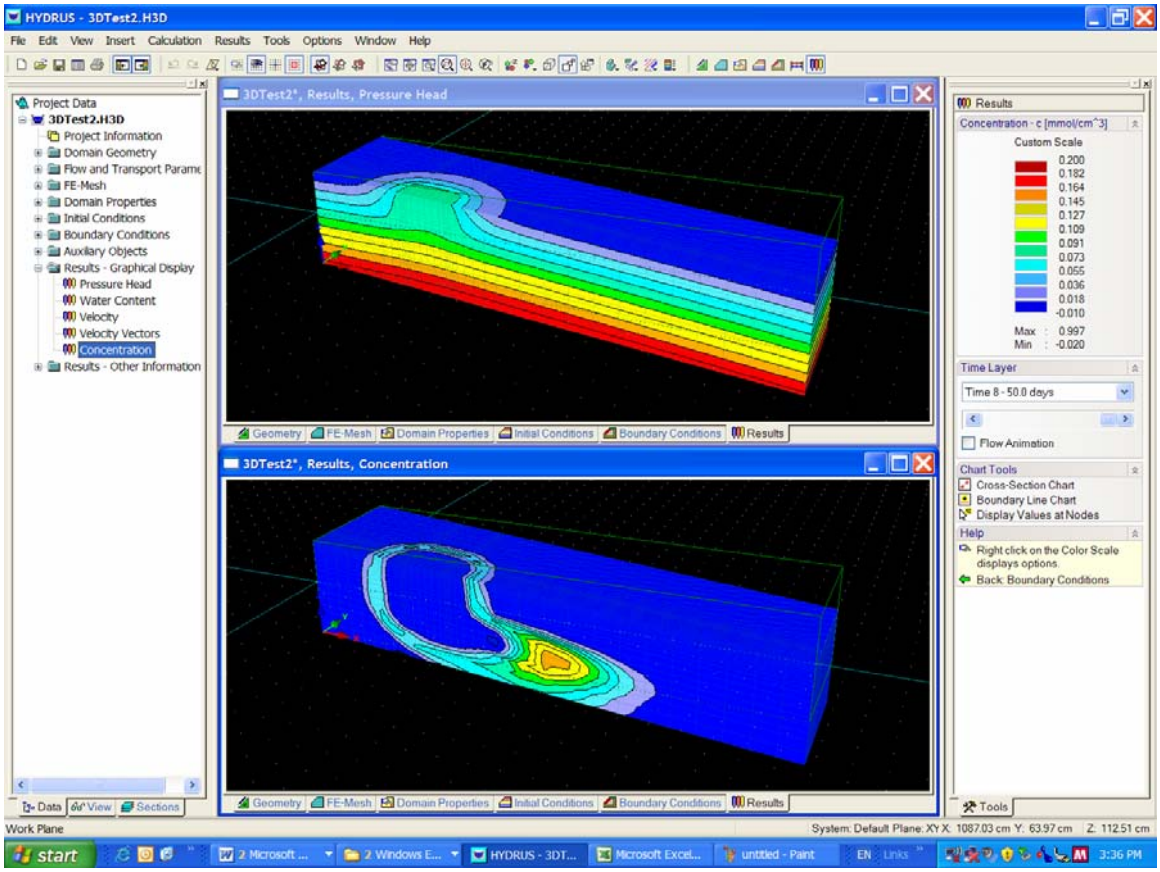
HYDRUS (2D/3D) Computer Session

Three-Dimensional Water Flow and Solute Transport



This tutorial considers water flow and solute transport in a simple three-dimensional transport domain. The transport domain is a relatively simple hexahedral domain with a slope in the X-direction. Dimensions of the transport domain are 1000 * 250 * 200 cm and there is a groundwater 100 cm below the soil surface. There is a source of water and contaminant at the soil surface. The problem is divided into two parts. In the first part, the geometry of the transport domain and its discretization is defined and initial and boundary conditions are specified. In the second part, final pressure head profile from the first run is imported as an initial condition, and pulse of solute is added into the surface source. The example thus again demonstrates how results of a previous simulation can be used in follow-up calculations with different boundary conditions or having additional features. Users will learn how to define a simple three-dimensional transport domain and how to use Sections when defining initial and boundary conditions. Users will also learn various ways of viewing transport domain and simulation results.

A. Three-Dimensional Water Flow

Project Manager (File->Project Manager)
Button "New"

New Project (or File->New Project)
Name: 3DTest1
Description: 3D HYDRUS short course example - water flow
Working Directory: Temporary – is deleted after closing the project
Button "Next"

Geometry Information (Edit->Domain Geometry->Geometry Information)
Type of Geometry: 3D-Layered
Domain Definition: Hexahedral
Units: cm
Initial Workspace: Xmin = 0 cm, Xmax = 1000 cm, Ymin = 0 cm, Ymax=250 cm, Zmin = 0 cm, Zmax=200 cm (to accommodate the transport domain)
Button "Next"

Hexahedral Domain Definition Information (Edit->Domain Geometry->Geometry Definition)
Dimension: Lx = 1000 cm, Ly = 250 cm, Lz = 200 cm
Slope: Alpha = - 5^{o}, Beta = 0
Button "Next"

Main Processes (Edit->Flow and Transport Parameters->Main Processes)
Check Box: Water Flow
Button "Next"

Time Information (Edit->Flow and Transport Parameters->Time Information)
Time Units: days
Final Time: 5
Initial Time Step: 0.0001
Minimum Time Step: 0.00001
Maximum Time Step: 5
Check Time-Variable Boundary Conditions
Number of Time-Variable Boundary Records = 1
Button "Next"

Output Information (Edit->Flow and Transport Parameters->Output Information)
Print Options:
Check T-Level Information
Check Screen Output
Check Press Enter at the End
Print Times: Count: 10
Update

Print Times: 0.25, 0.5, 0.75, 1, 1.5, 2, 2.5, 3, 4, 5
Button "Next"

Water Flow - Iteration Criteria (Edit->Flow and Transport Parameters->Water Flow Parameters->Output Information)

Leave default values as follows:
Maximum Number of Iterations: 10
Water Content Tolerance: 0.001
Pressure Head Tolerance: 1
Lower Optimal Iteration Range: 3
Upper Optimal Iteration Range: 7
Lower Time Step Multiplication Factor: 1.3
Upper Time Step Multiplication Factor: 0.7
Lower Limit of the Tension Interval: 0.0001
Upper Limit of the Tension Interval: 10000
Initial Condition: In the Pressure Head
Button "Next"

Water Flow - Soil Hydraulic Model (Edit->Flow and Transport Parameters->Water Flow Parameters ->Soil Hydraulic Model)

Radio button - van Genuchten-Mualem
Radio button - No hysteresis
Button "Next"

Water Flow - Soil Hydraulic Parameters (Edit->Flow and Transport Parameters->Water Flow Parameters ->Soil Hydraulic Parameters)

Leave default values for loam
Explore Catalog of Soil Hydraulic Properties and Neural Network Predictions
Button "Next"

Time-Variable Boundary Conditions (Edit->Flow and Transport Parameters->Variable Boundary Conditions)

Time = 5 d
Transp = 0
Var.H-2 = 100
Var.H-3 = 100
Button "Next"

Hexahedral Domain Spatial Discretization (Edit->FE-Mesh->FE-Mesh Parameters)

Horizontal Discretization in X

Count = 39

Entries in the x column: 0, 25, 50, 75, 100, 125, 150, 170, 185, 195, 200, 205, 210, 220, 235, 250, 265, 280, 290, 295, 300, 305, 315, 330, 350, 375, 400, 450, 500, 550, 600, 650, 700, 750, 800, 850, 900, 950, 1000

Horizontal Discretization in Y

Count = 18

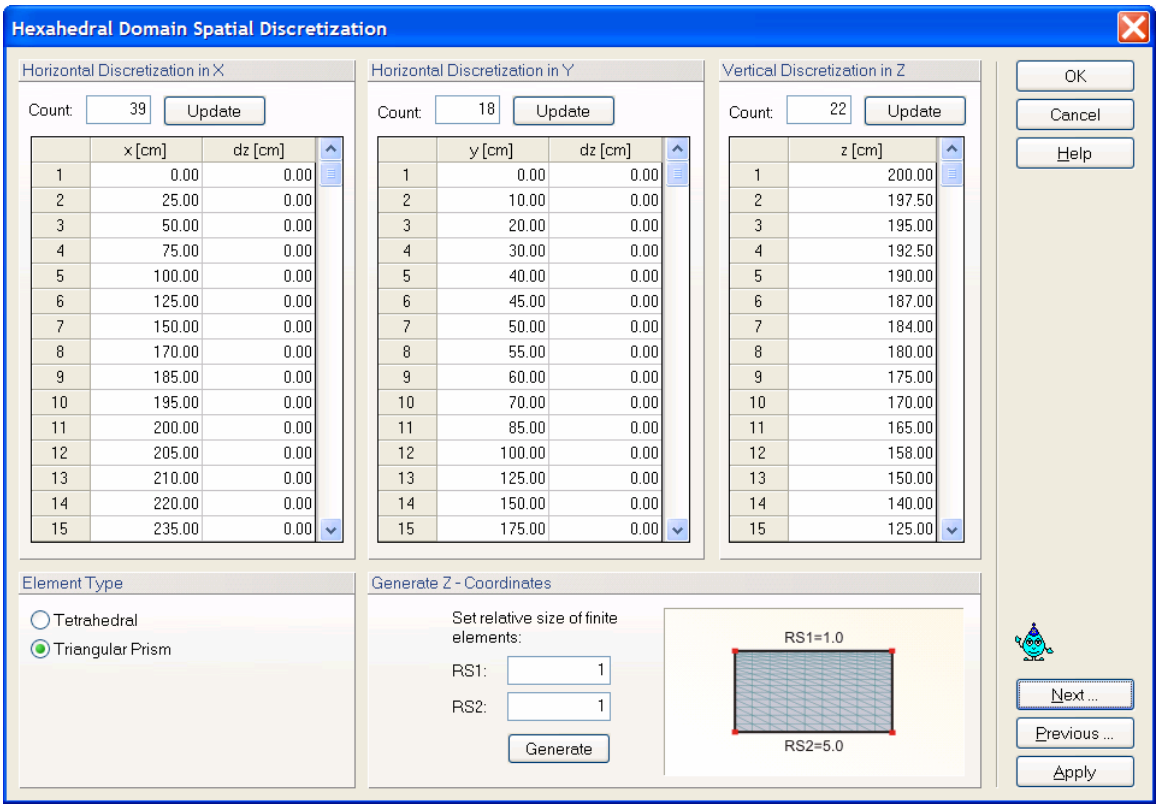
Entries in the y column: 0, 10, 20, 30, 40, 45, 50, 55, 60, 70, 85, 100, 125, 150, 175, 200, 225, 250.

Horizontal Discretization in Z

Count = 22

Entries in the z column: 200, 197.5, 195, 192.5, 190, 187, 184, 180, 175, 170, 165, 158, 150, 140, 125, 110, 95, 80, 65, 50, 25, 0

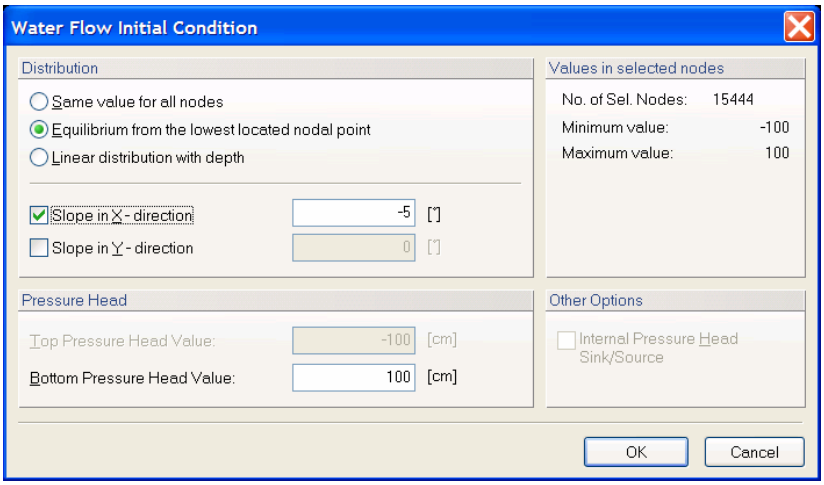
Button "Next"



Default Domain Properties (Edit->Domain Properties->Default Domain Properties)

Button "Next"

Water Flow Initial Conditions:
Select the entire transport domain.
Edit Bar: Click on Set Values
In the "Water Flow Initial Condition" dialog select:
Equilibrium from the lowest located nodal point
Slope in X – direction = -5^{o}
Bottom Pressure Head Value: 100 cm



Boundary conditions:

Tool Bar: View Commands ( ): In Y-direction (or from Menu: View->View in Direction->In Y-direction)

Tool Bar: Perspective view ( ) (or from Menu: View->Perspective)

Select the first column of nodes on the left and on the Edit Bar select "Variable Head 2" boundary condition.

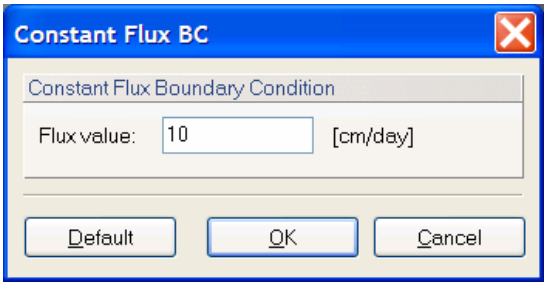
Select the last column of nodes on the right and on the Edit Bar select "Variable Head 3" boundary condition.

Navigator Bar: Select the "Section" Tab and select "D2_001 Mesh Layer, Z=200 cm" Section

Tool Bar: View Commands ( ): In Reverse Z-direction (or from Menu: View->View in Direction->In Reverse Z-direction).

Tool Bar: Zoom by Rectangle ( ) and zoom on area of approximately X=(150 - 350 cm) and Y=(0-100 cm)

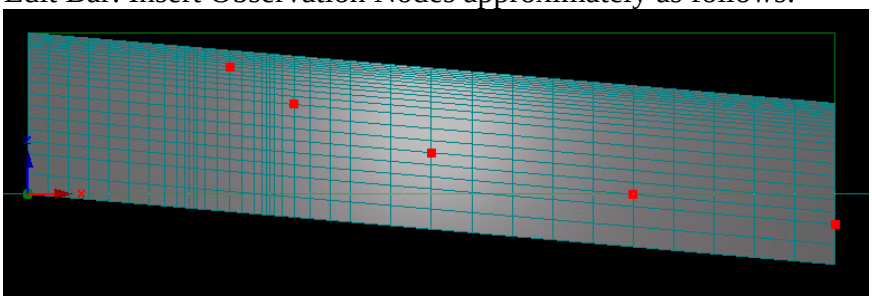
Select nodes between X=(200-300 cm) and Y=(0-50 cm).
From the Edit Bar select the "Constant Flux" boundary condition and in the "Constant Flux BC" dialog specify Flux value of 10 cm/d.



Observation Nodes (Tab Domain Properties or Insert->Domain Properties->Observation Nodes)
Navigator Bar: Select the "Section" Tab and select "D1_001 Shell" Section

Tool Bar: View Commands ( ): In Y-direction (or from Menu: View->View in Direction->In Y-direction)

Edit Bar: Insert Observation Nodes approximately as follows:



Menu: File->Save (or from Toolbar)

Menu: Calculation->Run HYDRUS (or from Toolbar)
(Execution time on 3 GHz PC – 169 s)

OUTPUT:
Results – Graphical Display: Pressure Heads (from the Navigator Bar, or Results->Display Quantity->Pressure Heads from menu)

Use Listbox *Time Layer* or *Slidebar* on the Edit Bar to view results for different print times
Check *Flow Animation*
Select different display modes using Options->Graph Type

Results – Graphical Display: Velocity Vectors (from the Navigator Bar, or Results->Display Quantity->Velocity Vectors from menu)

Results – Other Information: Observation Nodes (from the Navigator Bar, or Results->Observation Nodes from menu)

Results – Other Information: Mass Balance Information (from the Navigator Bar, or Results->Mass Balance Information from menu)

B. Three-Dimensional Water Flow and Solute Transport

Close the 3DTest1 Project (click Save Project at the Toolbar or File->Save)

Project Manager (File->Project Manager)
Select the Plume1 project
Button "Copy"
Name: 3DTest1
Description: 3D HYDRUS short course example - water flow and solute transport
Button "3DTest2"

Main Processes (Edit->Flow and Transport Parameters->Main Processes)
Check Box: Solute Transport
Button "Next"

Time Information (Edit->Flow and Transport Parameters->Time Information)
Final Time: 50
Initial Time Step: 0.01
Button "Next"

Output Information (Edit->Flow and Transport Parameters->Output Information)
Print Options:
Print Times: Count: 8
Update
Print Times: 1, 2, 5, 10, 20, 30, 40, 50
Button "OK"

Solute Transport - General Information (Edit->Flow and Transport Parameters->Solute Transport Parameters->General Information)
Pulse Duration = 5 days
Button "Next"

Solute Transport - Transport Parameters (Edit->Flow and Transport Parameters->Solute Transport Parameters-> Solute Transport Parameters)
Disp.L = 10
Disp.T = 1
Button "Next"

Time-Variable Boundary Conditions (Edit->Flow and Transport Parameters->Variable Boundary Conditions)
Time = 50 d

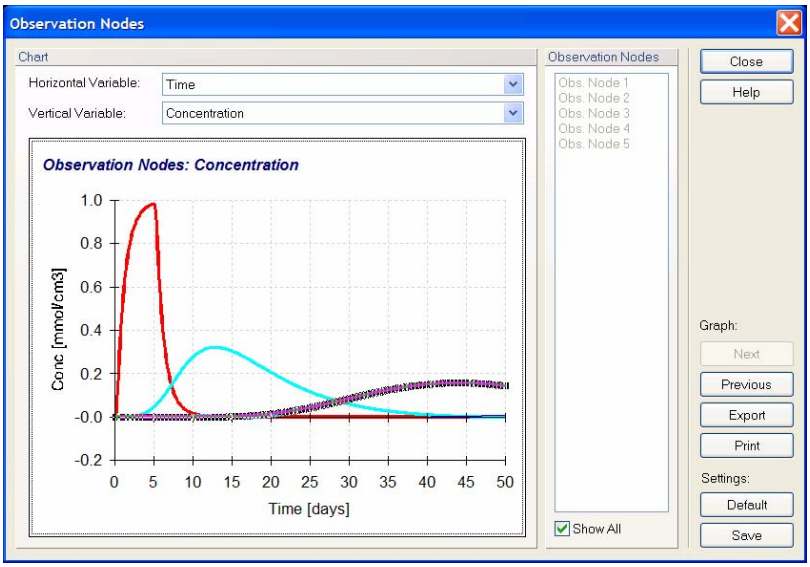
Specify Initial Condition:
Import the final pressure head profile from Source1 as the initial condition for Source1

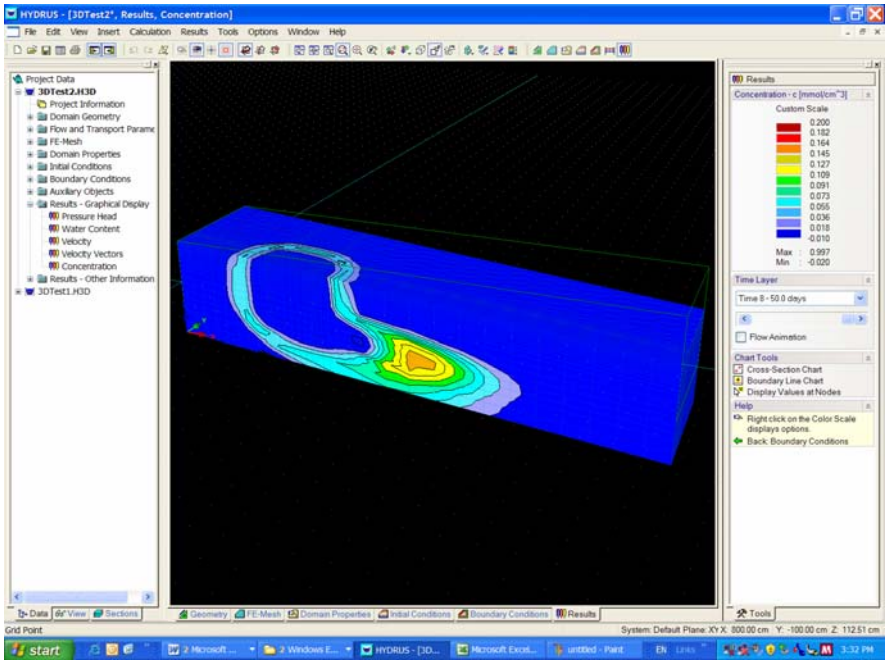
(Edit->Initial Conditions->Import)
Find project 3DTest1
Select Pressure Head and click OK

Menu: File->Save (or from Toolbar)

Menu: Calculation->Run HYDRUS (or from Toolbar)
(Execution time on 3 GHz PC – 10 min)

Check out various output options





Spatial Discretization

X	Y	Z
0	0	200
25	10	197.5
50	20	195
75	30	192.5
100	40	190
125	45	187
150	50	184
170	55	180
185	60	175
195	70	170
200	85	165
205	100	158
210	125	150
220	150	140
235	175	125
250	200	110
265	225	95
280	250	80
290		65
295		50
300		25
305		0
315		
330		
350		
375		
400		
450		
500		
550		
600		
650		
700		
750		
800		
850		
900		
950		
1000		