EUROAQUAE
Euro Hydro-Informatics and Water Management

Workshops

Physically-Based Catchment Modelling

Tutorial 4 : Groundwater flow modelling of the lower Var alluvial aquifer

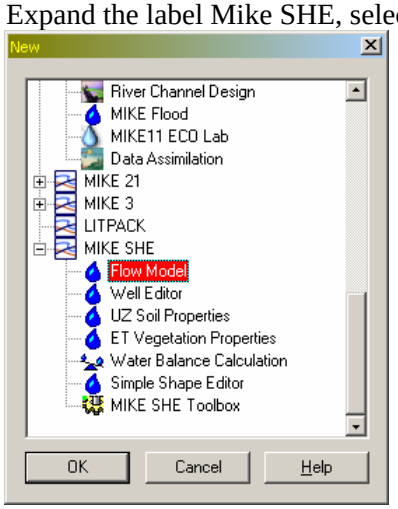
E. Makovsky, University of Budapest
Budapest University of Technology and Economics, Muregyetem rkp. , 3 Budapest - Hungary

V. Guinot, University of Montpellier
Maison des Sciences de l'Eau, 34095 Montpellier Cedex 5, France

1. Start Mike SHE going in the « Start Menu », Programs\DHI Software\Mike SHE\Mike SHE. The DHI software interface called “Mike Zero” will be then started.

2. For creating the setup file, in the “File” menu select “New”. A window is opened, which gives you the possibility to specify the type of DHI software you want to use.

3. Expand the label Mike SHE, select “Flow model” and click OK.



4. Save the configuration model in the directory SHE\Model Var under the name of Var-aquifer.SHE.

5. First of all, you have to specify the simulation parameters. If you click on “Simulation specifications”, the menu that appears will allow you to choose the flow modules that you want to include in the simulation.

5.1 Tick the boxes “Water movement (WM)”, “Saturated flow (SZ)”, Overland flow (OL) and Rivers and lakes (OC).

5.2 In the sub-menu “simulation period” you can define the start date and the end date of the simulation. For the start date set 1999/01/01 00:00; for the end date 2000/01/01 00:00.

5.3 Go to the sub-menu “Time step control”, and set the following values:

Initial time step:	0.01 hrs
Max allowed OL time step:	2 hrs
Max allowed SZ time step:	2 hrs
Mike 11 time step:	0.25 min
Increment rate:	0.05
Max precipitation depth per time step:	1 mm
Max infiltration amount per time step:	1 mm

Input precipitation rate requiring its own time step: 0.1 mm/hr

5.4 In the sub-menu called “OL Computational Control” set the following iteration parameter values:

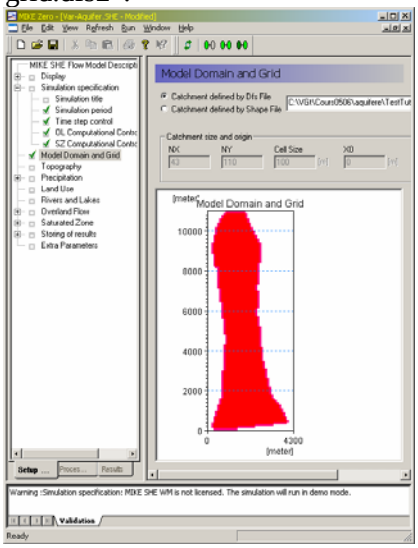
Max number of iterations:	20
Iteration stop criteria:	0.00001

Water depth threshold: 0.0001

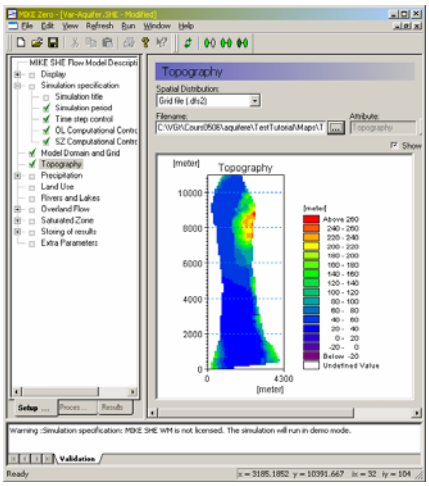
5.5 Go to the sub-menu “SZ Computational Control Parameters”. Set the following parameters:

Max no. of iterations:	20
Max head change per iteration:	0.0001
Max residual error:	0.001
Saturated thickness threshold:	0.05

6. In the menu “Model domain and grid” you can define the topology of the model by means of a map of the catchment area. The grid file you need can be selected by a click on the triple dot button. When the file selection menu appears, select the file “Maps\Model domain and grid.dfs2”.



7. Go to the menu “Topography”. In the window “Spatial distribution”, set “Grid file (dfs2)”. The topography data file can be selected in the same way: click on the triple dotted button, and select “Maps\Topography.dfs2. If the map is loaded, you can see an overview of the map in the window.



8. Go to the menu “Precipitation”. Check that the option “Include snowmelt” is not activated.

8.1 Go to the sub-menu “Precipitation rate”. In the window “Spatial distribution”, select the option “Station based”. For “Data type”, select “Grid codes”, and load the file containing the location of the rainfall station from Model Var\Maps\Precipitation station.dfs2.

8.2 The software will automatically create as many sub-menus as there are precipitation station codes. In the menu “Temporal distribution” select “Time varying”, and go to the sub-menu Grid code=1. Load the data file called “DailyRain” situated in Model Var\Time, which contains the precipitation data for the specified period.

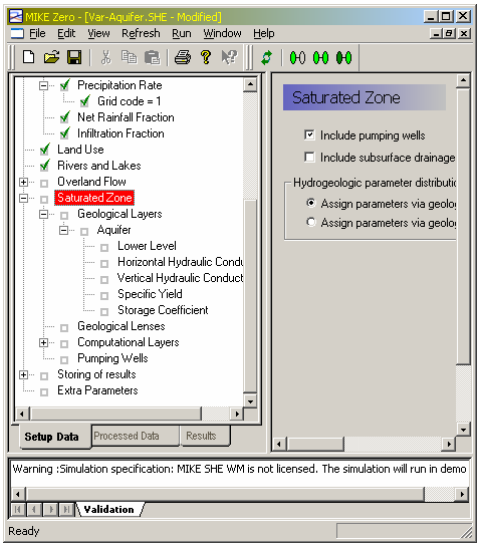
8.3 Go to the sub-menu net rainfall fraction. Choose the option “uniform” and set the value of 1.

8.4 Go to the sub-menu “Infiltration fraction”. Set the infiltration factor to a uniform value of 1.

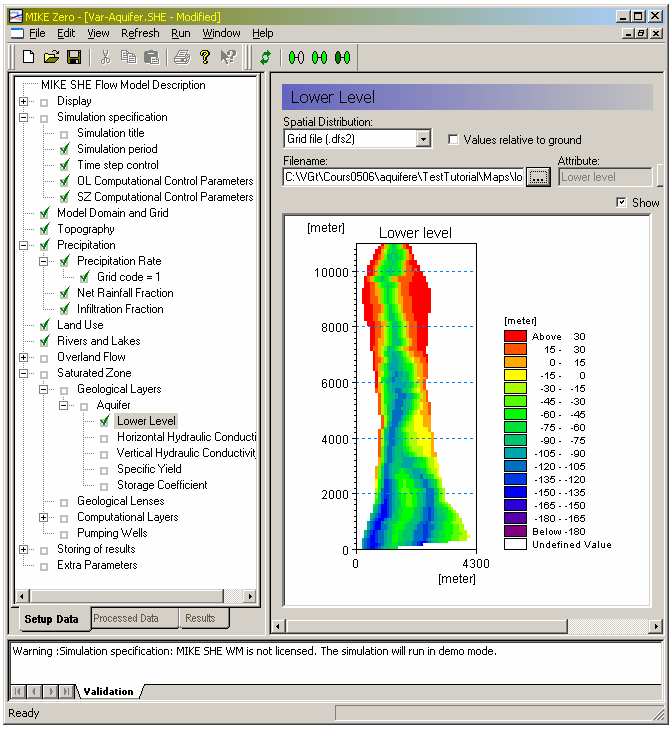
9. Go to the menu “Land use” and check that the box “Paved areas” is not activated.

10. Go to the menu “Saturated zone”. Tick the box “Include pumping wells”, which will permit you to define the parameters of the water catchments established on the model area. Then expand the sub-menu “Geological layers”.

10.1 Now you have another sub-menu, called “Aquifer”. Expand it. In this model, there will be only one geological layer, the elevations of the base level of which are described by the Lower level data file, the top level corresponds to the topography..



10.2 Go to “Lower level”. Here you can specify the elevations of the lower level of the aquifer, in form of a map. For the “Spatial distribution”, select “Grid file”, click on the triple dotted button and load the file “Maps/Lower level.dfs2”. The overview of the geological layer will then appear in the window.



10.2 In the sub-menu “Horizontal hydraulic conductivity”, set the uniform value of 0.004 m/s.

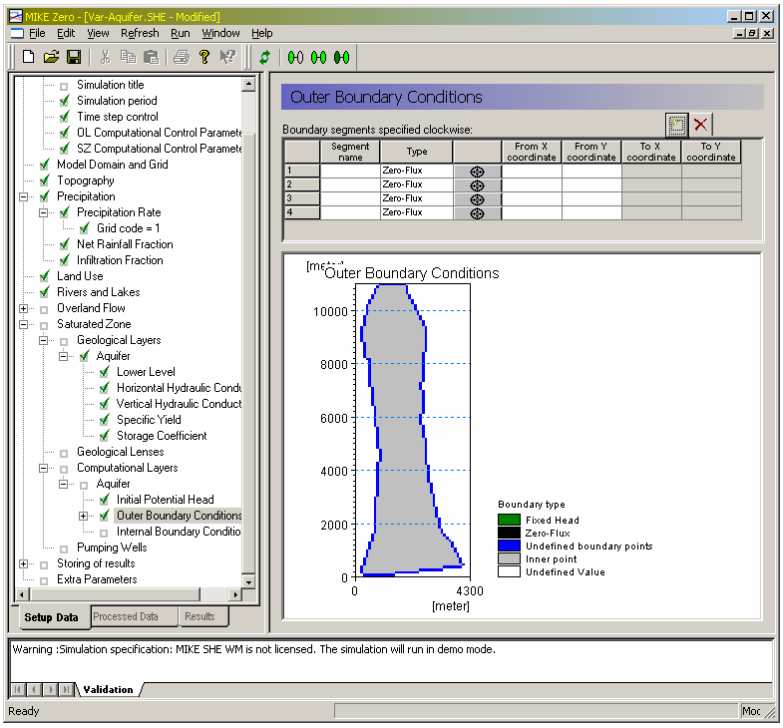
10.3 Go to the sub-menu “Vertical hydraulic conductivity”, and set the uniform value of 0.0002 m/s.

10.4 In the sub-menu “Specific yield”, set 0.05, et in the next sub-menu, “Storage coefficient”, set 0.2 1/m. Make sure that the menu “Geological lenses” is empty.

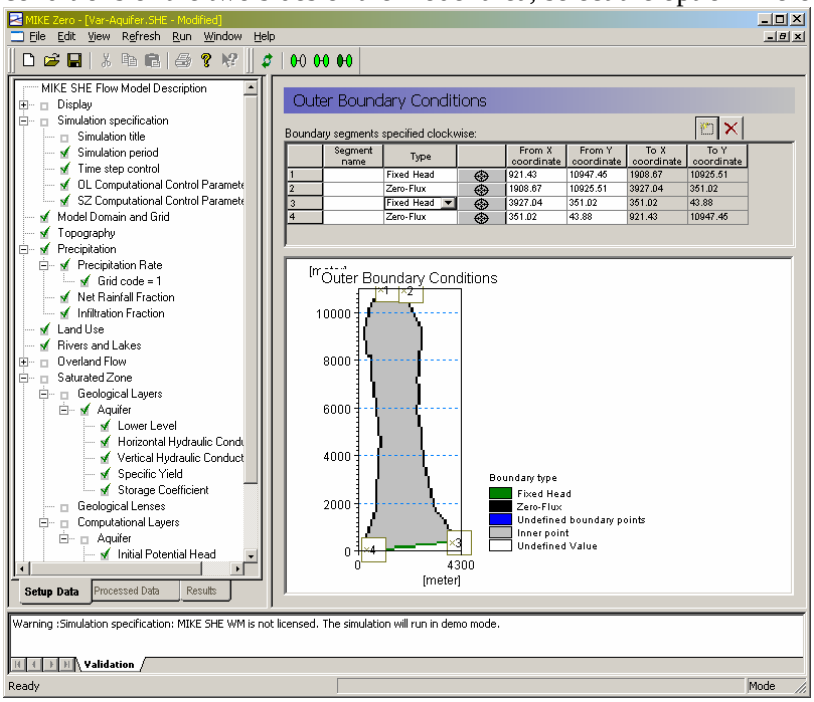
10.5 Go to “Computational layers”, and check if the value of “Minimum layer thickness” is 0.5 m. The type of vertical discretisation should be “Defined by geological layers”.

10.6 Expand the sub-menu “Aquifer” in “Computational layers”, and go to “Initial potential head”. Here you can define the head elevation in saturated zone. For the spatial distribution choose “Grid file (dfs2)”, and load the data file “Maps/Hfinal”.

10.7 In the sub-menu “Outer boundary conditions”, add four lines to the window “Boundary segments specified clockwise” by the help of the button “Insert” situated in the upper right corner of the window. The interface should look as follows



The four segments will represent the two sides, the upstream and the downstream boundary conditions of the catchment area. You can define the segments by clicking on the button that represents a cross-hairs, and indicating the corner points of the model area by a click on the left button of the mouse on the map. For the upstream and downstream boundary conditions, choose the type of “Fixed head”, for the boundary conditions of the two sides of the model area, select the option “Zero flux”.



The software will automatically create as many sub-menus as there are different boundary condition segments. Select the sub-menu of the upstream boundary condition. For the Data type, set “fixed(specified)”, and set the value of 53 m. For the

downstream boundary conditions, you should use the same data type, and set the value of 0 m.

10.8 Go to the “Internal boundary conditions”, and make sure that the option “None” is indicated.

10.9 Go to the sub-menu “Pumping wells”, and load the data file “Wells/captage.wel”. This file contains the locations of the pumping wells as well as the pumping time series. If you tick the box “Show well location on the map”, a window appears that presents the locations of the pumping wells on the model area.

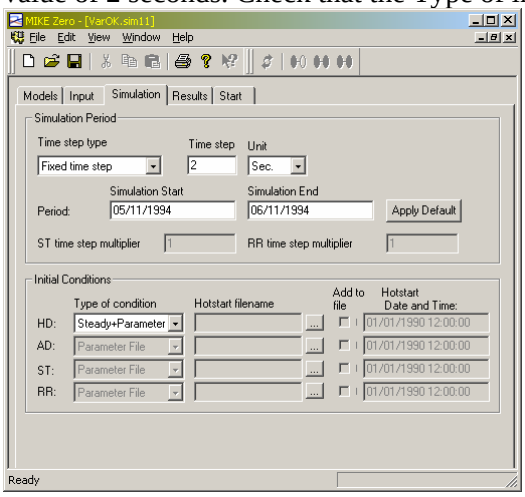
11. Go to the “Rivers and lakes”. The parameters of the river network have to be defined by the Mike11 simulation file “RiverNetwork/VarOK.sim11”.

12. By a click on the “Edit” button beside the river simulation file name, the Mike11 interface that allows you to create the description and the simulation parameters of the river network.

12.1 “Models” thumbnail: The “hydrodynamic” and the “unsteady” options should be activated.

12.2 “Input” thumbnail: Here you can specify the file of which the river network data can be loaded. The data are filed in the directory Model Var\RiverNetwork. For the Network, you have to load “Network.nwk11”, for Cross sections “Xsections.xns11”, for the boundary data “Var-OK.bnd11”, and for HD parameters “Var-OK.HD11”.

12.3 “Simulation” thumbnail: For the simulation start, set 05/11/1994. The simulation end will be 06/11/1994. Choose the option “Fixed time step” and for time step, set the value of 2 seconds. Check that the Type of initial condition is “Steady+parameter”.



12.4 “Results” thumbnail: Here you can specify a name for the results file. The extension of the file must be res.11, and must be saved in the directory RiverNetwork. Then specify the storing frequency, setting the value of 10 in the window.

12.5 “Start” thumbnail: If you have specified correctly the parameters, “Run parameters” and “HD parameters” the lights should be green now. If this is the case, close the Mike11 interface. If it is not, please check that all of the previous steps have been completed correctly.

13. Go to the "Overland flow" menu. Activate the option "Full contact in entire catchment". In the sub-menu "Manning number" Type the value of 10 $m^{1/3}/s$. In the two other sub-menus, "Detention storage" and "Initial water depth", the spatial distribution should be uniform and the values should be equal to 0.

14. For specifying result storage, go to "Storing of results". In the sub-menu "Grid series output", specify the following storing time steps:
    "Prec., OL,OC": 2 hrs
    "SZ": 6 hrs
    "SZ flow": 6 hrs
    Then tick the boxes "depth to phreatic surface", head elevation in saturated zone", and "SZ exchange flow" with river".

15. Compile the model and start the simulation.