Visual HEIFLOW Tutorial

February, 2018

Prepared by

Yong Tian & Yi Zheng

tiany@sustc.edu.cn

zhengy@sustc.edu.cn

School of Environmental Science & Engineering

Southern University of Science and Technology

Version 1.0.0

1 Overview

Visual HEIFLOW (VHF) is a comprehensive graphical data processing and modeling system for integrated surface water-groundwater modeling. This tutorial provides a step-by-step process of creating and running a HEIFLOW model. Necessary files for creating the model accompany this document. These files include shapfiles, raster, meteorological data, and observed hydrological and groundwater data. This documentation can assist with creating a HEIFLOW model for a basin.

The workflow of establishing a HEIFLOW model and running it in VHF is illustrated by Figure 1-1.

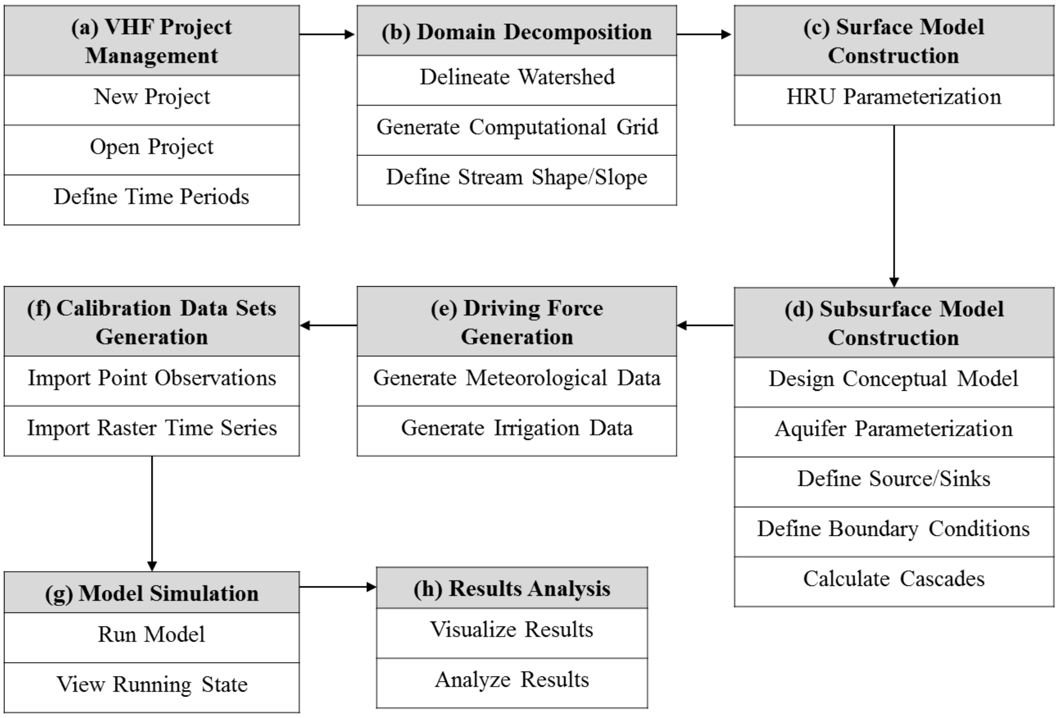

Figure 1-1. The overall workflow of using VHF to establish and run an integrated

2 Installing VHF software

2.1 System requirements

VHF requires:

Hardware:

- Personal computer using 64-bit (x64) processor, which runs a 2 GHz or faster
- 2 GB RAM (64-bit)
- 200 MB available disk space

Software:

- Microsoft Windows 7 or Windows 10 operating system
- Microsoft .NET framework 4.5 or higher
- Microsoft Access 2013 or higher

2.2 Installing VHF

1) In the VHF Install folder, double click the “setup.exe” program.
2) Click “OK”. The next dialog will begin the installation. Click “Next”, then choose an installation folder.

2.3 Overview of VHF interface

When you start VHF for the first time, you will see the interface as shown in Figure 2-1. The basic elements of the interface consist of the ribbon and a set of child windows. The ribbon is the primary interface for working in VHF, it help you quickly find the commands that you need to complete a task. The ribbon is organized into tabs. Tabs organize the commands into logical groups.

Some common operations in the main interface are described as follows:

1) Show or collapse the ribbon: at the top-left corner, click the Minimize/Expand the Ribbon icon . Double click on arbitrary Tab can also show or collapse the ribbon.
2) Dock windows: some child windows in VHF are dockable. You can dock,

show or hide these windows, e.g., Legend Window, Map Window, etc. Taking the Legend Window as an example, after you grab its title bar, you should see a translucent blue rectangle that represents the new location of the Legend (Figure 2-2). Drag your cursor over to the left arrow/chevron (either one). The translucent rectangle will change size to show its docked position. Release, and the window should be docked.

3) Hide/Show windows: all child windows of VHF could be shown or closed. For dockable windows (e.g., Legend Window), click the Auto Hide icon  to hide the window and click the Close icon  to close the window. For the child window that can't be docked, click the icon to close the window. If a window has been closed, switch to the View Tab, click corresponding icon to show the window once again.

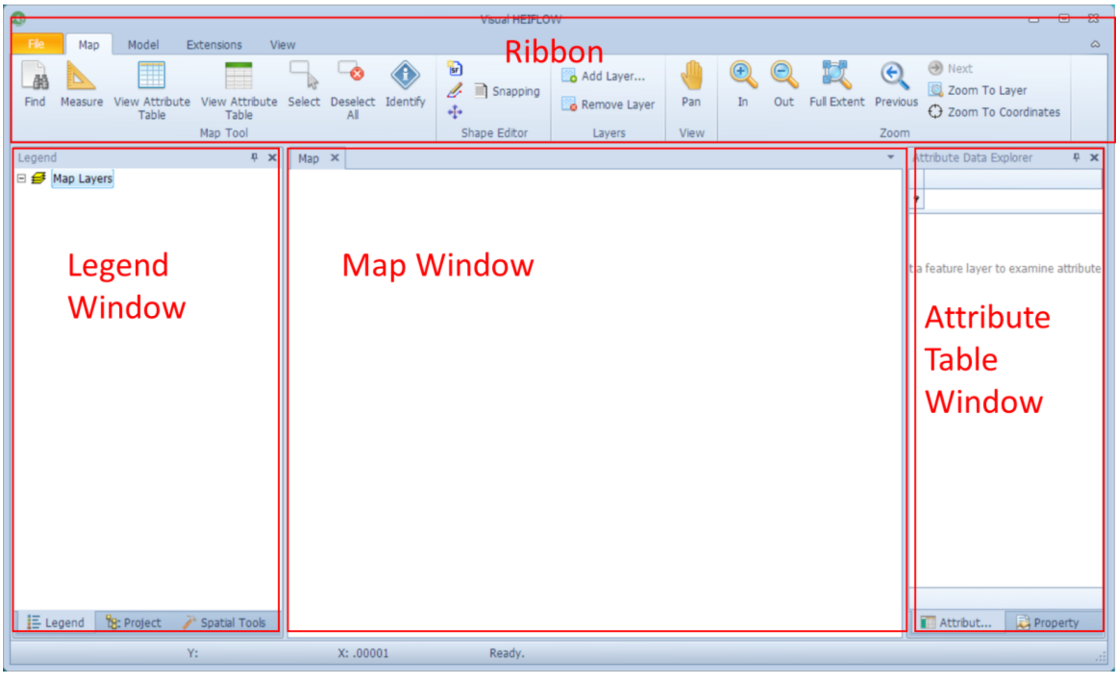

Figure 2-1. Main user interface of VHF.

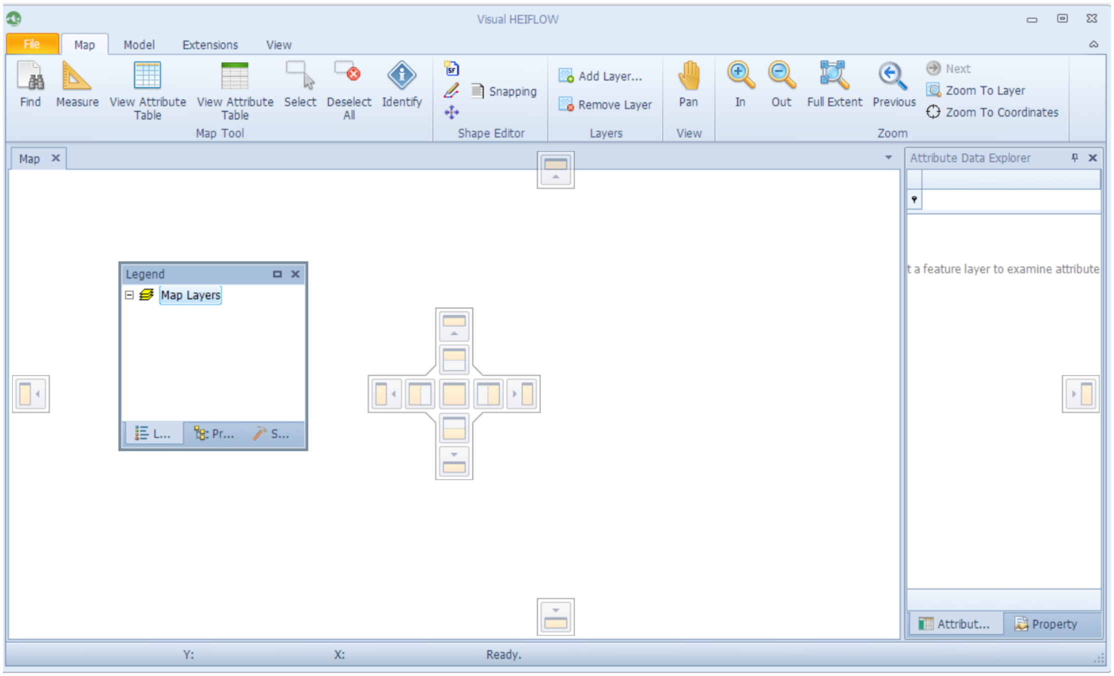

Figure 2-2. Dock legend window.

Some commonly used child windows are described as follows:

1) Legend: it lists all the layers on the map and shows what the features in each layer represent. The check box next to each layer indicates whether its display is currently turned on or off. The order of layers within the Legend Window specifies their drawing order in the data frame.
2) Project Explorer: it is the major means to manage a HEIFLOW project. A HEIFLOW project contains a reference to a HEIFLOW model on your disk. It organizes organize and manage all components of the HEIFLOW model as logical collections.
3) Model Toolbox: it contains a powerful set of tools that perform the most various operations on the Data Cube matrix.
4) Spatial Tools: it provides a set of tools that perform fundamental GIS operations.

3 Example Study Area

3.1 Overview of the study area

In this tutorial, we will model the Miho catchment (Figure 3-1) located in Korea. The Miho catchment is in the headwaters of the Geum River basinhich and drains the central part of the Korean peninsula. The population of the Miho catchment was approximately one million in 2011 according to the Water Management Information System (WAMIS, http://www.wamis.go.kr). The elevation ranges from 7 to 631 m. The length of the Miho stream is approximately 97 km, and the annual areal discharge is approximately 600 mm.

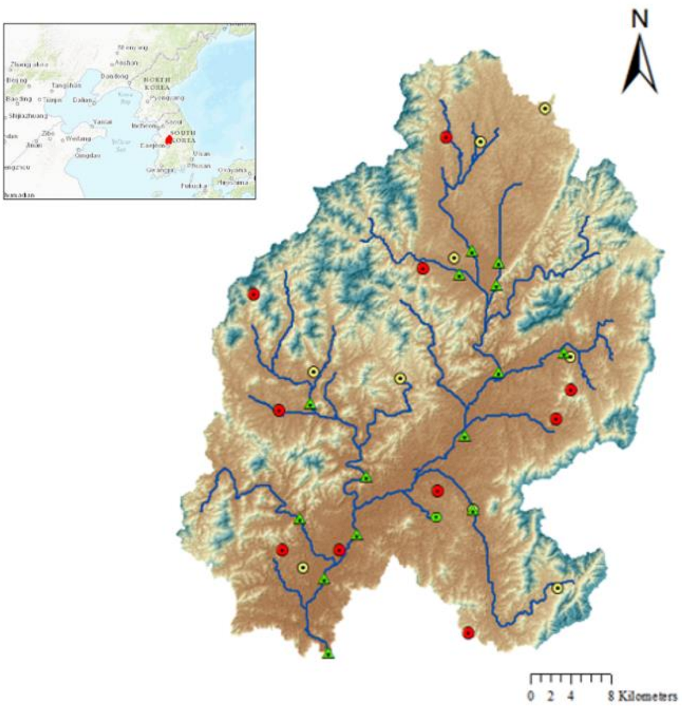

Figure 3-1. Example study area- The MIHO catchment in Korea

3.2 Source Data

All necessary data used for the modeling accompany this document. The data can be downloaded from the website https://github.com/DeepHydro/VHF/releases. The data are organized into four folders (Figure 3-2). The data were provided by the Water Management Information System (WAMIS), Korea Meteorological

Administration (KMA) and Groundwater Information Service.

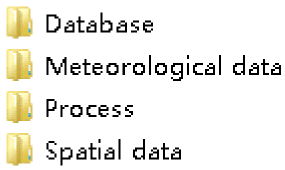

Figure 3-2. Folders in the tutorial data.

- Database folder: contains a Access2013 file named “MIHO.accdb”;
- Meteorological data folder: contains two csv files. The “Meteorological data.csv” contains daily meteorological data (maximum temperature, minimum temperature and potential ET) observed at the Cheongju weather station. The “rainfall7.csv” contains precipitation observed at 7 rainfall gauging stations in the study area. The meteorological data covers the period from 2004 to 2014.
- Process folder: this folder contains a series of csv file used to assign model parameters.
- Spatial data folder: contains GIS data used for the modeling.

4 Domain decomposition

4.1 Project Setup

The first step in using VHF is to set up a project so that necessary folders and databases are created to store all the data.

1. Select File -> New Project from the main menu bar. In the prompted dialog entitled “New Project”, input “Miho” in the textbox next to Name and specify a Location (“C:\Users\Administrator\Documents”) to store all files needed for HEIFLOW model run (Figure 4-1).

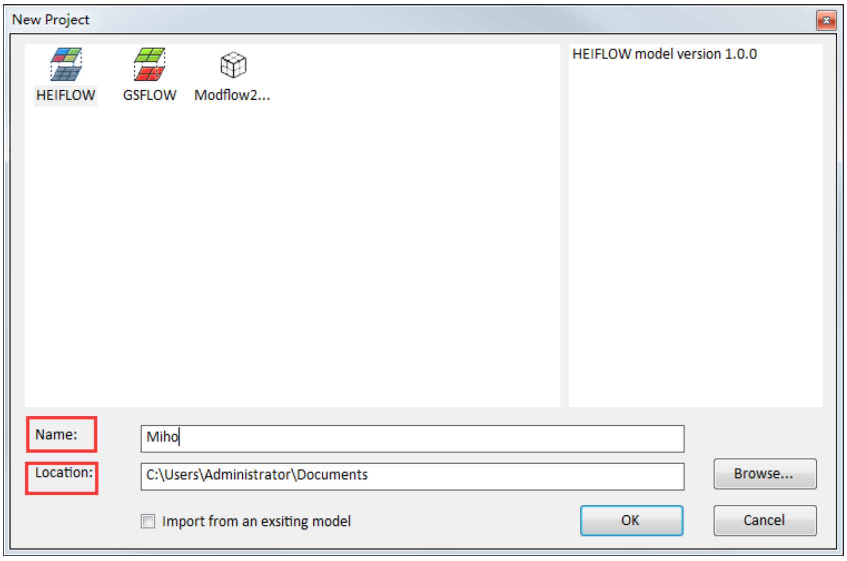

Figure 4-1. New project dialog

2. Click OK after you setup a project. Then a VHF project will be created under the Location directory (Figure 4-2).

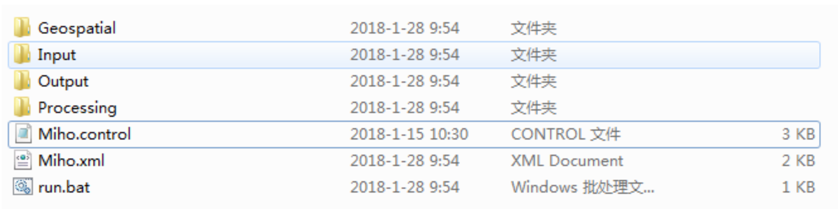

Figure 4-2. The structure of VHF project directory

3. Select Map >> Add Layer…, navigate to the Spatial Data directory in the Tutorial folder in the prompted dialog, select “dem.tif” and then click Open. The DEM will be added to the map, and the main window of VHF looks like Figure 4-3.

4. Repeat previous step to add “soil_type.tif” and “landcover.tif”.

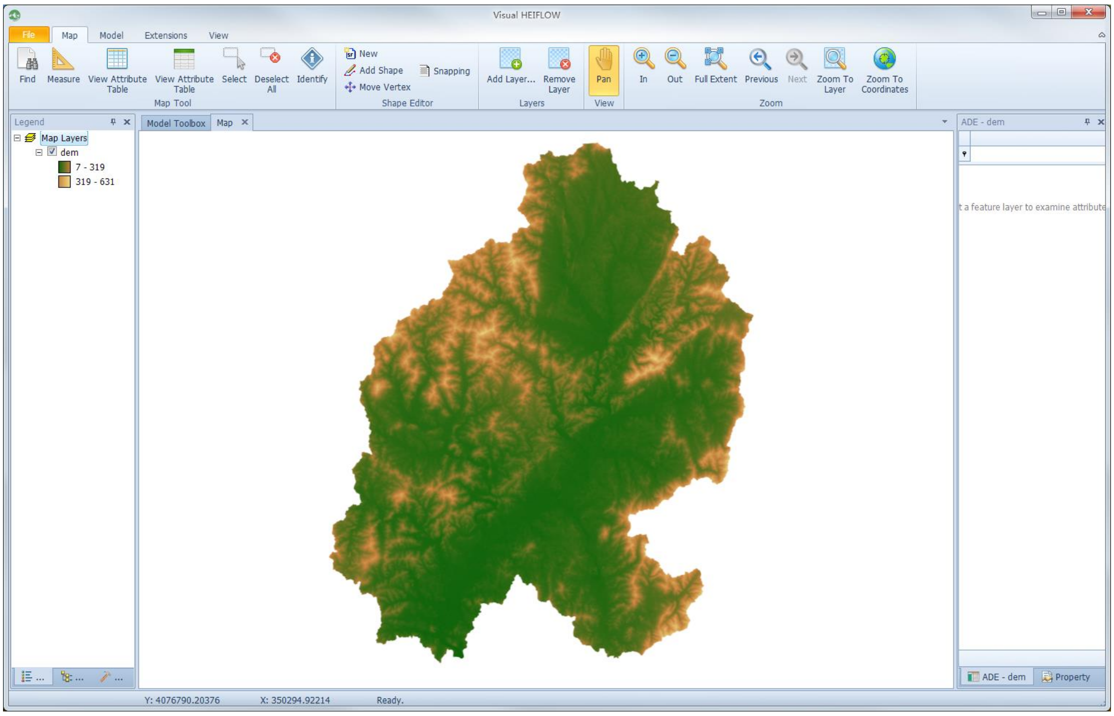

Figure 4-3. DEM of Miho catchment shown in the Map window.

4.2 Watershed Delineation

1. Select Model >> Watershed Delineation, the Automatic Watershed Delineation tool will appear (Figure 4-4).

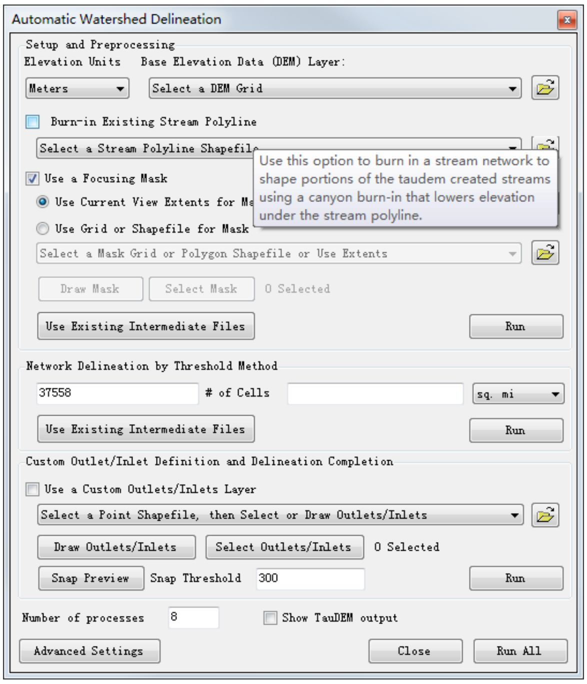

Figure 4-4. Dialog of the Automatic Watershed Delineation tool

2. Select “dem” from the dropdown list that is below the label “Base Elevation Data (DEM) layer”. Click Run All button at the bottom of the dialog.

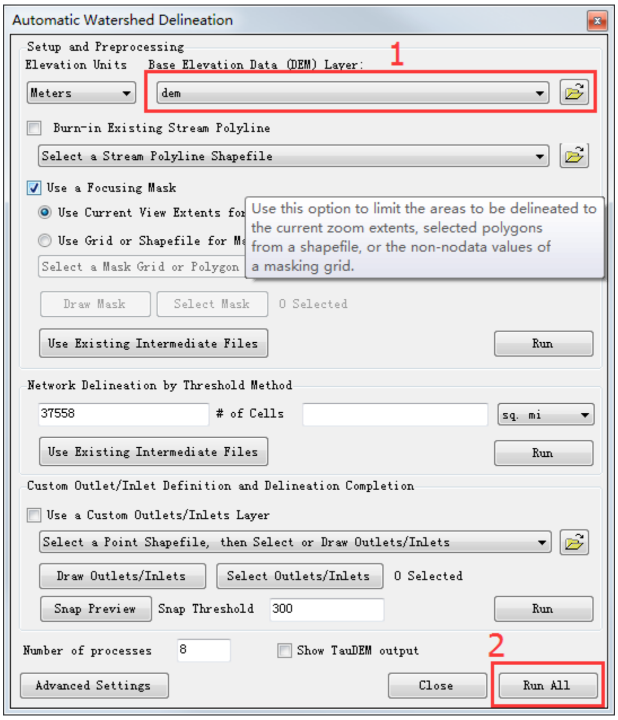

Figure 4-5. Performing delineation.

3. The tool starts to perform delineation, and the tool dialog looks like Figure 4-6.

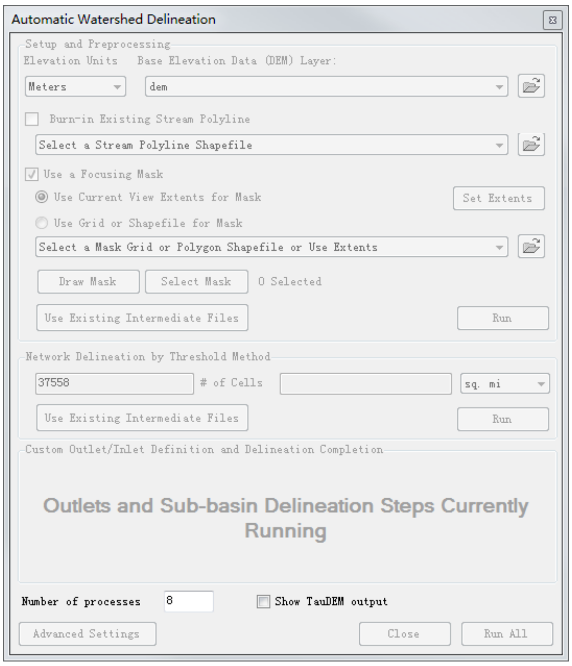

Figure 4-6. Waiting interface when performing delineation.

4. Once the delineation is completed, a dialog may appear to ask you select a definition option for projection. Click OK in the dialog. Three layers will be added to the map: “demnet”, “demw” and “dem_basin”. They represent stream network, watersheds and basin boundary, respectively.
5. You can adjust layer order in the Legend Panel by drag-drop operations.
6. Right click on the “demw” layer, select Properties in the prompted context menu, then the Layer Properties Dialog will appear.
7. Set layer display style as indicated by Figure 4-7.

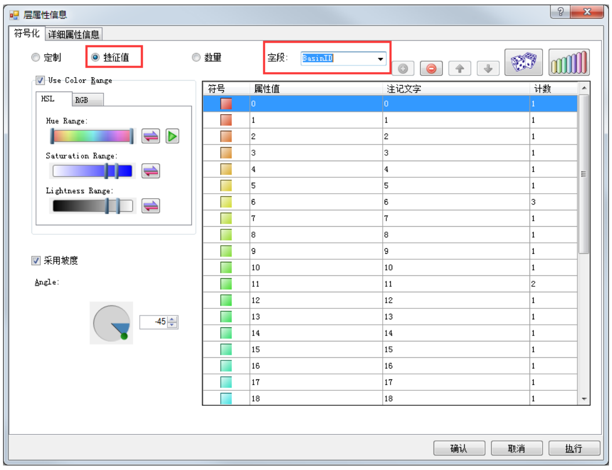

Figure 4-7. Layer Properties Dialog

8. Now the map looks like:

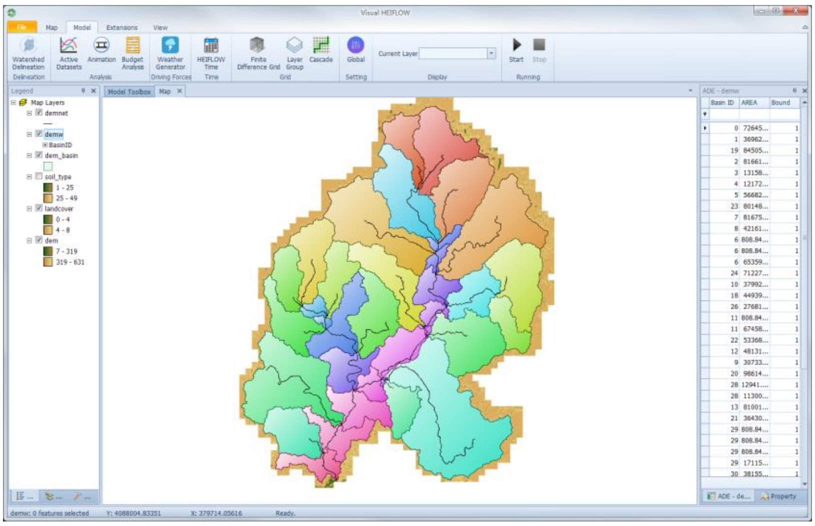

Figure 4-8. Displaying effect after setting style of the Watershed layer.

9. You can set display style of other layer as previous step.

4.3 Time and Spatial Dissertation

1. Select Model >> HEIFLOW Time, a dialog entitled Model Time appears. Select Start as 2004-1-1 and End as 2014-12-31 respectively.

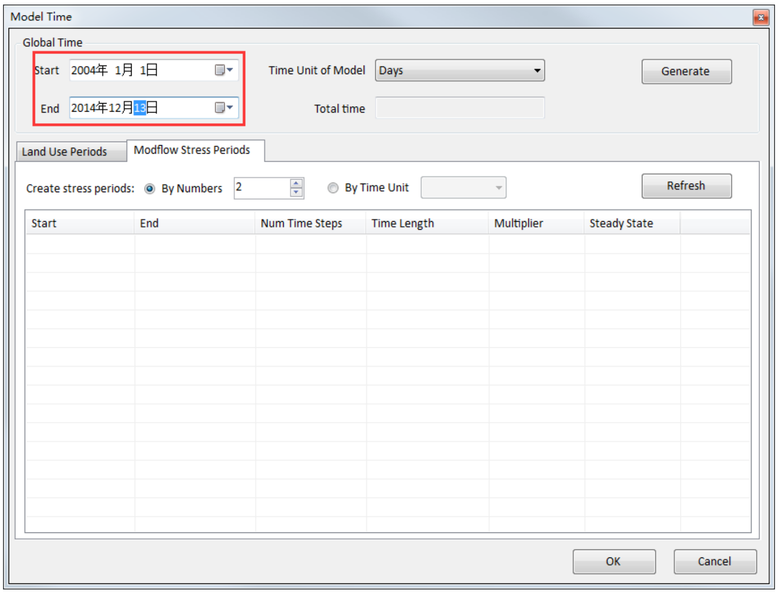

Figure 4-9. Dialog used to setting modeling time.

2. Click the button Generate at the upper-right, then Modflow Stress Periods will automatically generated, as shown in Figure 4-10.

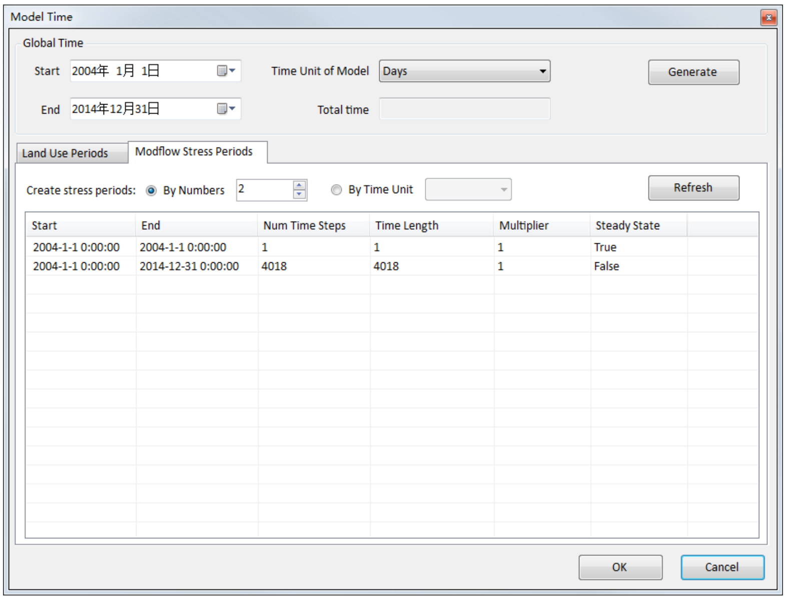

Figure 4-10. The generated global Model Time and Modflow stress periods.

3. Select Model -> Layer Group in the Ribbon, then a dialog entitled “Aquifer Layer Group” appears:

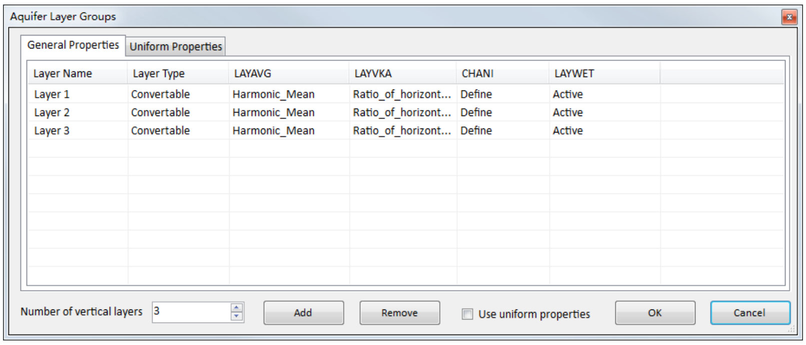

Figure 4-11. Aquifer Layer Group Dialog.

4. Set proper Layer Type and LAYWET by double-clicking corresponding cells for Layer 2 and Layer 3 (Figure 4-12):

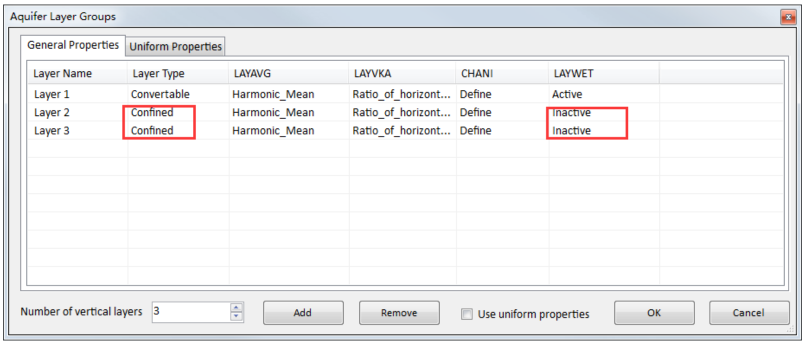

Figure 4-12. Setting aquifer layer properties

5. Select Model >> Finite Difference Grid, then a dialog entitled "Create Finite Difference Grid" appears:

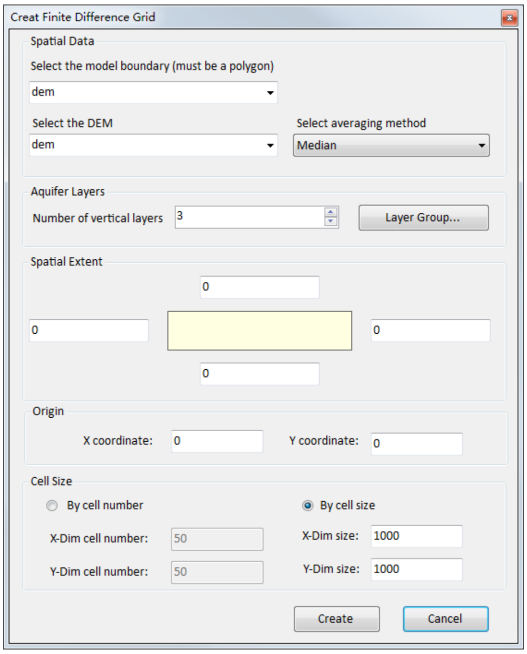

Figure 4-13. Create Finite Difference Grid dialog

6. In the above dialog, select "dem_basin" as the model boundary, and "dem"

as the DEM. Then click Create.

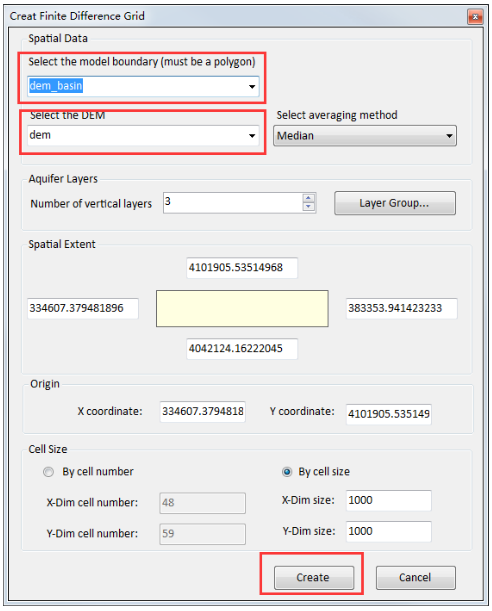

Figure 4-14. Settings of creating finite difference grid.

7. Once the grid is created, two new layers “Grid” and “Centroid” will added to the map as shown in Figure 4-15.

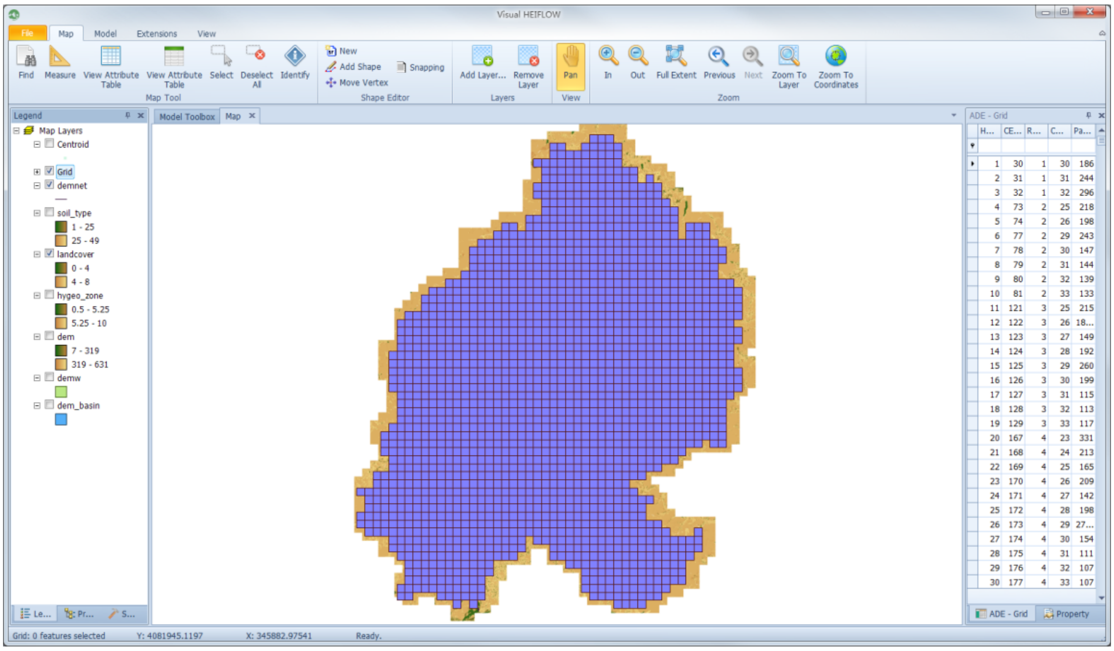

Figure 4-15. Generated model grid and cell centroids.

8. Select Project Explorer Panel in the left side of the main window, locate to Modflow-DIS-Elevation, right click on Top Elevation, and click Map View… (Figure 4-16). Then the map looks like Figure 4-17.

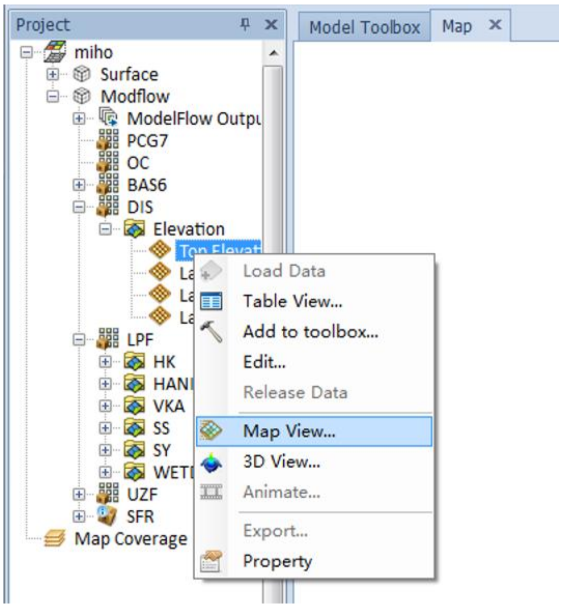

Figure 4-16. Map View context menu

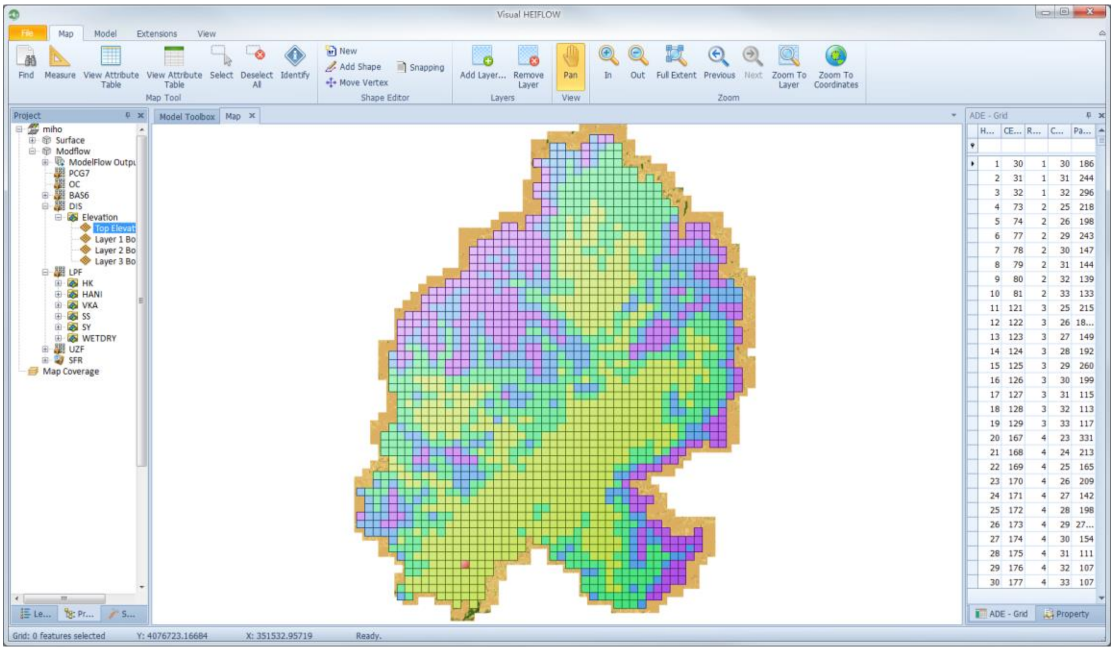

Figure 4-17. Map View of Top Elevation

9. In the context menu shown in Figure 4-16. Map View context menu, click 3D View, then a dialog entitled “3D View” appears:

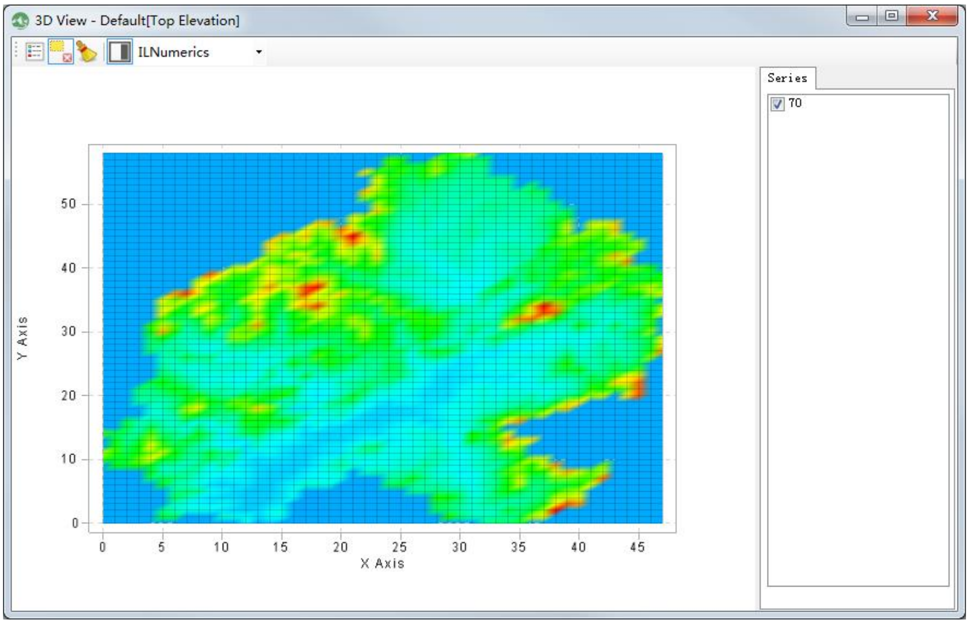

Figure 4-18. 3D View dialog

10. In the above dialog, left click and drag on the chart to change perspective (Figure 4-19. 3D View of Top Elevation):

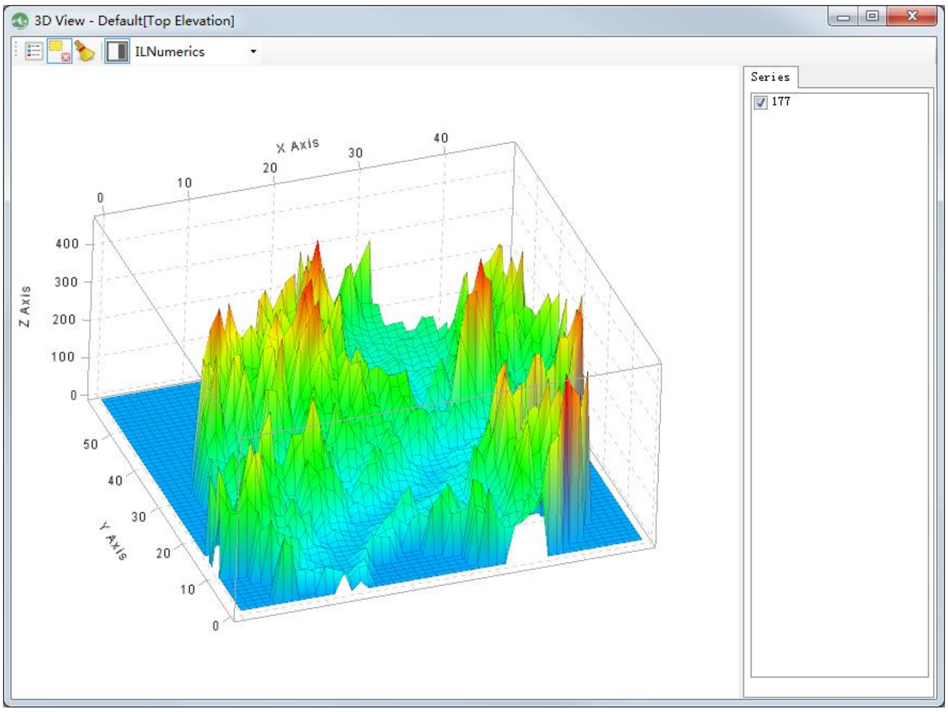

Figure 4-19. 3D View of Top Elevation

11. Select File >> Save Project, a dialog will appear to show the progress of saving.

5 Construction of Surface and Subsurface Models

Before demonstrating how to construct surface and subsurface models, it is helpful to introduce the Project Explorer (错误!未找到引用源。), which is the major means to manage an IHM, such as to add or remove a package. Many commands can be executed on the packages by right click menus in the project explorer.

After the domain decomposition, all necessary packages and default values of parameters in every package will be automatically generated. Project Explorer organizes all these packages, parameters and other modeling components through a hierarchy structure. As shown in 错误!未找到引用源。, the hierarchy is comprised of Integrated Model -> Surface / Subsurface Model -> Package -> Variables Folder -> Variable.

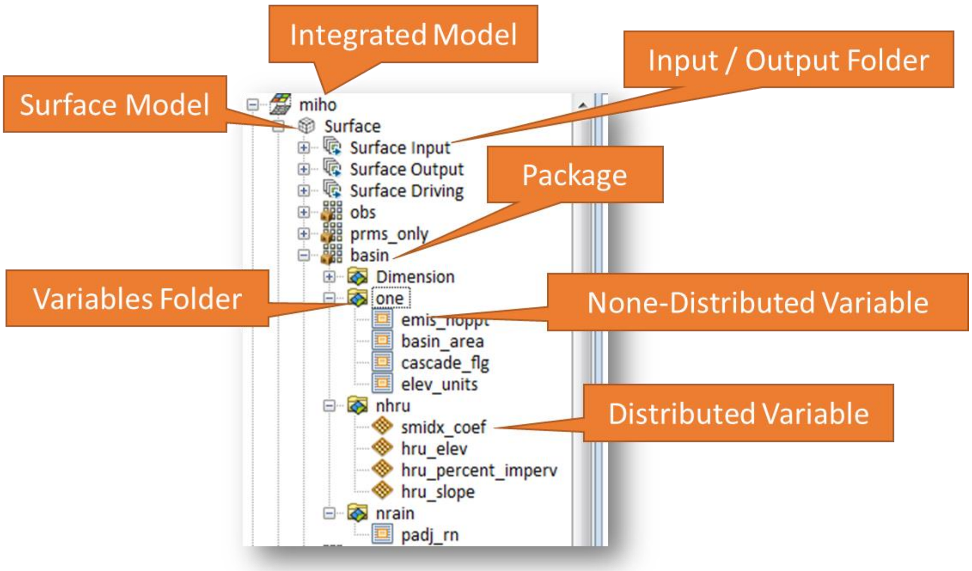

Figure 5-1. Structure of Project Explorer.

5.1 Surface model construction

5.1.1 Overview of Dimension-Variables and Parameters

The specification defined by PRMS Parameter File is used to store surface model parameters. The PRMS Parameter File includes a Dimension Section and a Parameter Section. The dimensions section is used to define the size of dimensions that are used to allocate memory for parameters and variables required by the model. The parameters section is used to specify dimensions, data types and values of each parameter. Table 5.1 presents portion of dimension-variables. Note that in VHF values of all the dimension-variables are automatically determined and should not been changed by users.

Table 5.1 Dimension-variables specified in the dimensions section of the PRMS Parameter File

Variable name	Definition	Default value
	Spatial dimensions	
ngw	Number of PRMS ground-water reservoirs (used in PRMS-only simulations)	1
ngwcell	Number of MODFLOW finite-difference cells in a layer (includes active and inactive cells)	0
nhru	Number of HRUs	1
nhrucell	Number of unique intersections between gravity reservoirs in PRMS soil zone and MODFLOW finite-difference cells	0
nreach	Number of stream reaches on all stream segments	0
nsegment	Number of stream segments	0
nsfres	Number of on-channel detainment reservoirs (used in PRMS-only simulations)	0
nssr	Number of PRMS subsurface reservoirs (must be specified equal to nhru)	1
	Time-series input data dimensions	
nevap	Number of measurement stations that measure pan evaporation	0
nform	Number of input columns in PRMS Data File used to specify form of precipitation (0 if no form data, 1 if form data)	0
nobs	Number of streamflow-gaging stations	0
nrain	Number of measurement stations that measure precipitation	1
nsol	Number of measurement stations that measure solar radiation	0
ntemp	Number of measurement stations that measure air temperature	1
	Computation dimensions	
mxnsos	Maximum number of table values for computing storage in and flow from detention reservoirs using Puls routing (PRMS-only simulations)	0
ncascade	Number of cascade paths associated with HRUs	0
ncascdgw	Number of cascade paths associated with PRMS ground-water reservoirs	0
ndepl	Number of snow-depletion curves used for snowmelt calculations	1
ndeplval	Number of snow-depletion values for each snow-depletion curve	ndepl*11
	Fixed dimensions	
ndays	Maximum number of days in a year	366
nlapse	Number of lapse rates in the x, y, and z directions (used by module xyz_dist)	3
nmonths	Number of months in a year	12
one	A constant	1

VHF groups all the dimension-variables and parameters into different packages. Then in each package, the parameters are furthered grouped based on their dimension definitions. Taking the “basin” package as an example (Figure 5-2), it contains 7 dimension-variables (i.e., nhru, nssr, nobjfunc, mxnsos, nmonths, one and nhrucell) and 9 parameters. The 9 parameters are grouped into 3 categories based on their dimensions, e.g., the smidx_coef, hru_elev, hru_percent_imperv and hru_slope have the same dimension nhru, and thus they are in the same category entitled nrhu.

In Project Explorer, the distributed parameters (or variables) are presented using the icon, and the none-distributed parameters(or variables) are presented using the icon.

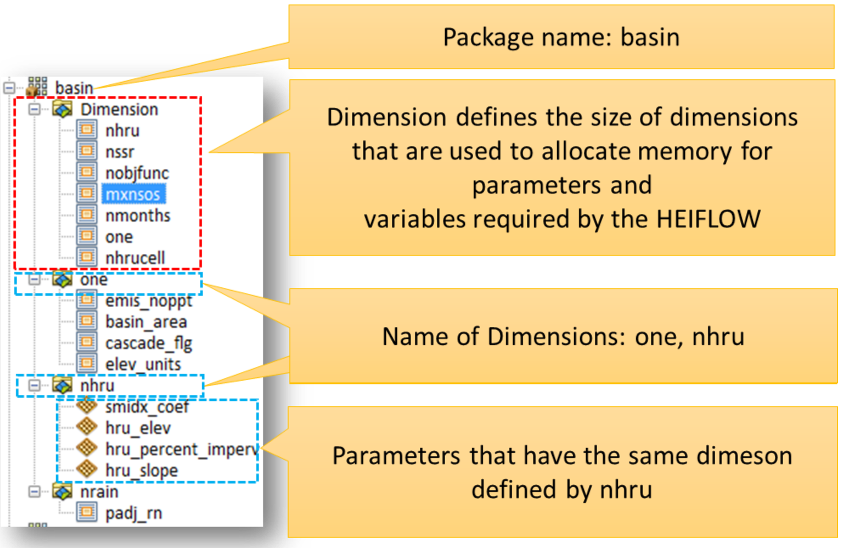

Figure 5-2. Presentation style of surface model's package.

Double-click on a parameter will prompt a floating panel entitled "Table View" (Figure 5-3). Values of the parameter is displayed in the table. Users are able to directly modify the parameter values in the table. Right click on a parameter and select Properties, the Property Window will be shown (Figure 5-3), and metadata about the parameter will be shown in the Property Window. The metadata includes value range (maximum and minimum), default value, units and descriptions.

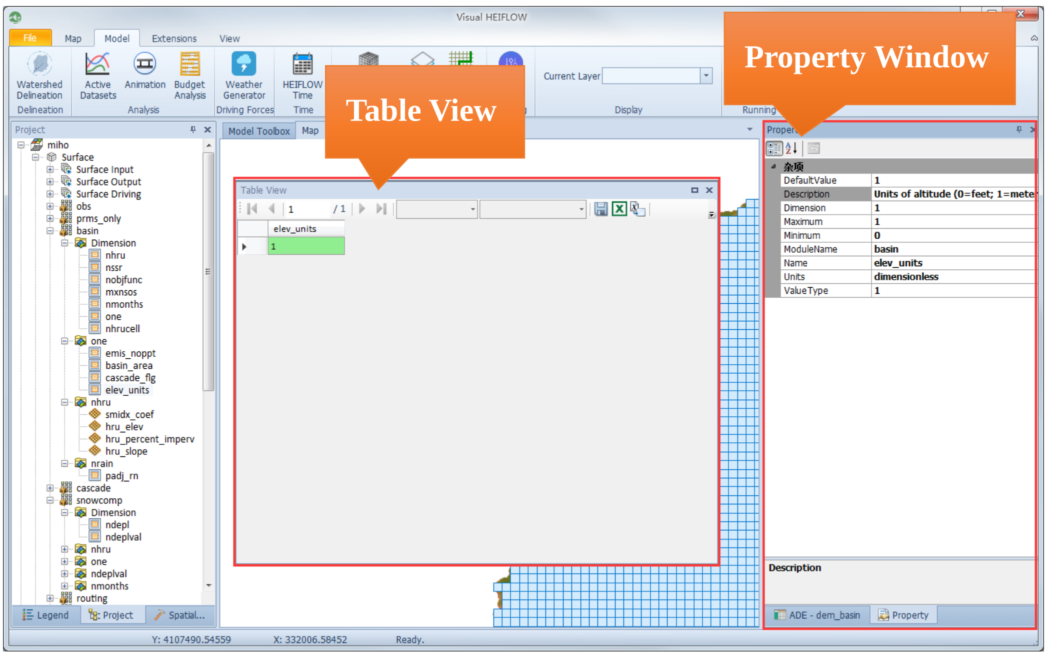

Figure 5-3. Table View and Property Window

5.1.2 Set Distributed-Parameters Values

VHF uses lookup table and GIS overlay operations to automatically assign parameter values to HRUs. The following illustrates how to set parameter values of the soilzone, intcp and basin packages.

1) Select Map -> Add Layer…, add landcover.tif and soil_type.tif into the map;
2) In the Project panel, right click on the Map Coverage, and select New Coverage… from the prompted context menu:

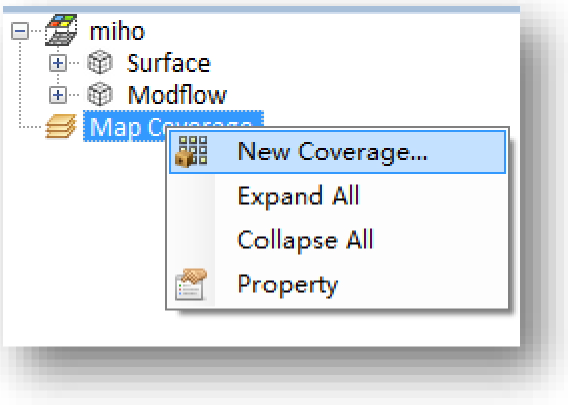

Figure 5-4. Create New Coverage.

3) In the prompted window entitled "Coverage Setup", input "soilzone_cov" in the textbox named Coverage Name, select "soil_type" from the Map Layer dropdown list, and select soilzone from the Packages dropdown list (Figure 5-5). Then all the parameters belonging to soilzone package will be displayed in the checklist that bellows the Areal Properties:

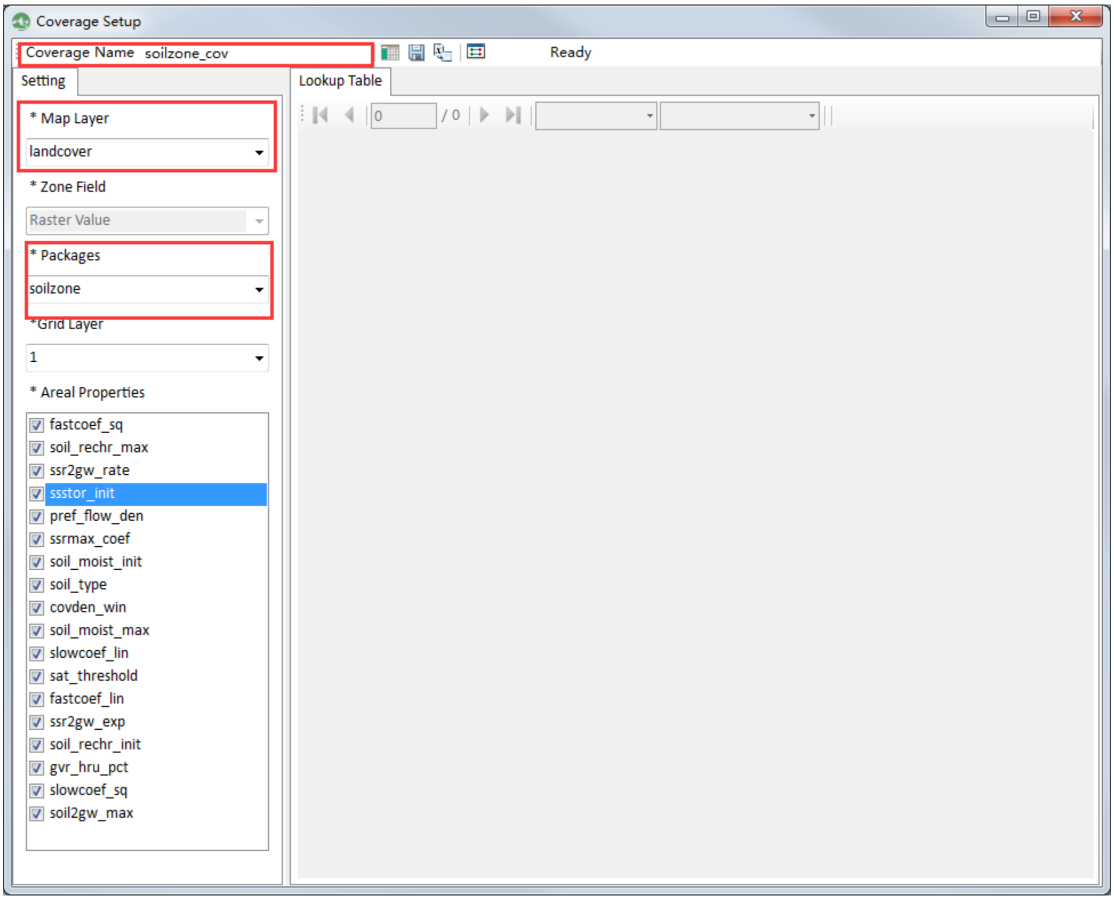

Figure 5-5. Create New Coverage for the soilzone package

4) Click “Create Lookup Table” in the toolbar (Figure 5-6), then a lookup table that relates the soilzone parameters to the “soil_type” map layer is created and displayed in the data table:

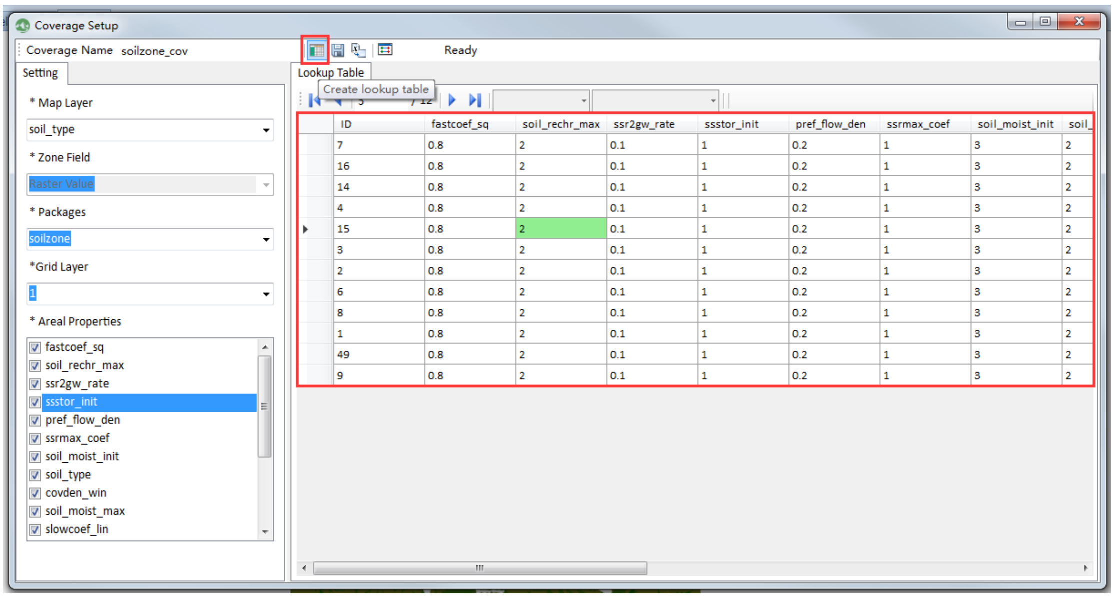

Figure 5-6. Create lookup table

5) In above figure, the first column of the lookup table lists unique values of the selected map layer (i.e., soil_type), and the rest columns list default values of the related parameters. You can directly modify the lookup table in the above interface or import from external file. This tutorial provides a prepared lookup table for the soilzone package. Click the button in the toolbar, and select the “soil zone.csv” file located in the Process Folder provided in the tutoral source data. Then the lookup stored in the csv file is imported and displayed in the table:

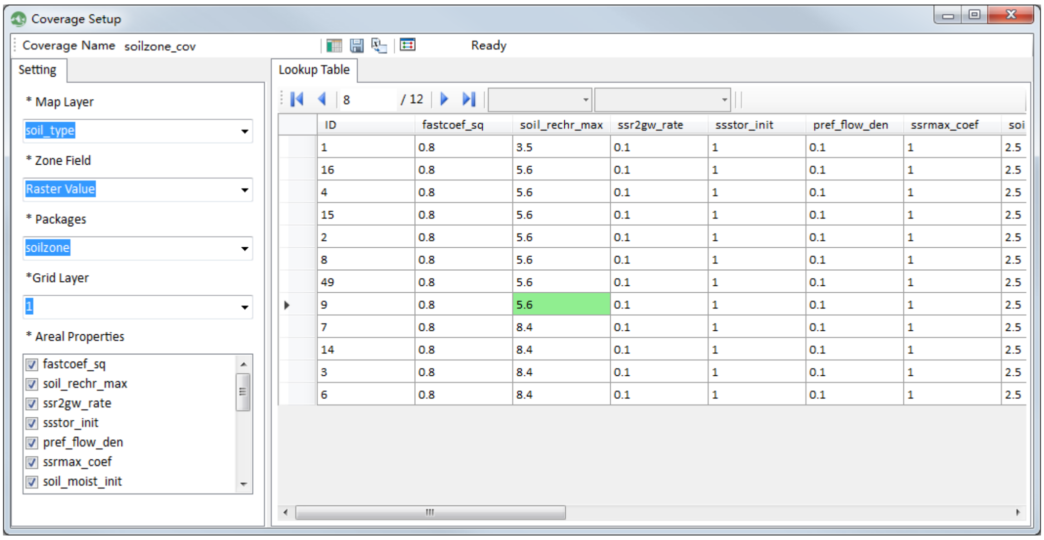

Figure 5-7. Imported lookup table

6) Click Save button in the toolbar, then click the Run parameterization button .

7) Close the Coverage Setup window.

8) In the Project Explorer, navigate to soil_type parameter, right click on it and select “Map View…” from the prompted context menu, then the “Grid” layer in the map looks like Figure 5-8. Adjust displaying order of map layers in the Legend Panel if needed.

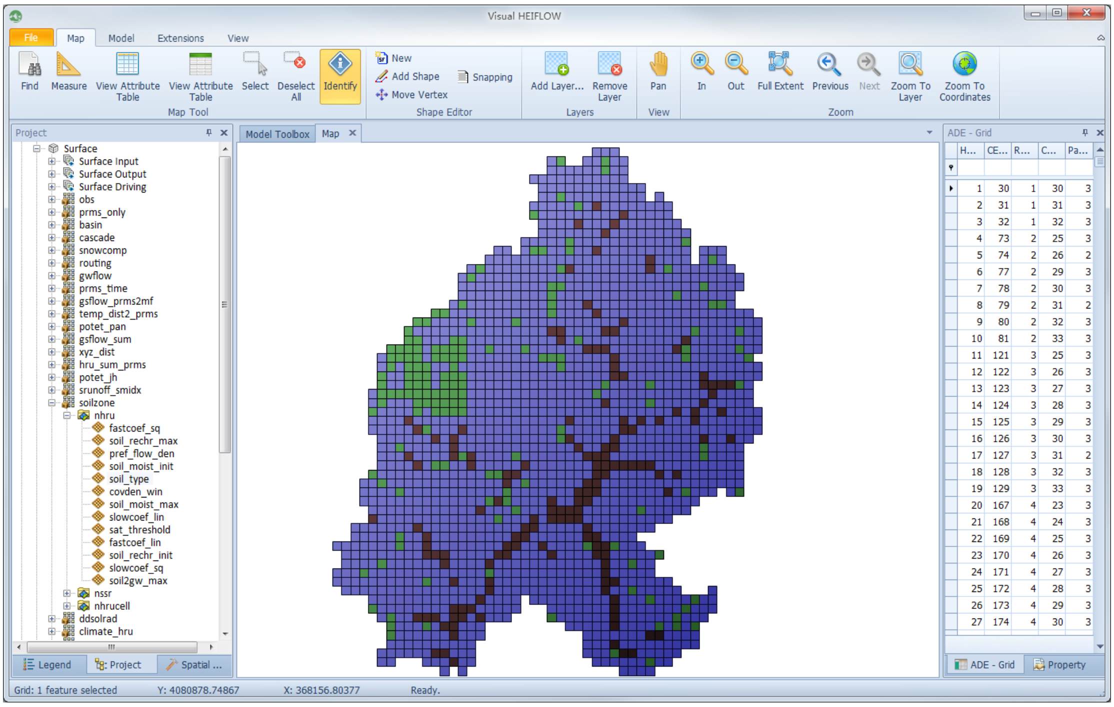

Figure 5-8. Map View of the soil_type parameter

9) Double-click on soil_type parameter, the Table View presents values of this parameter (Figure 5-9):

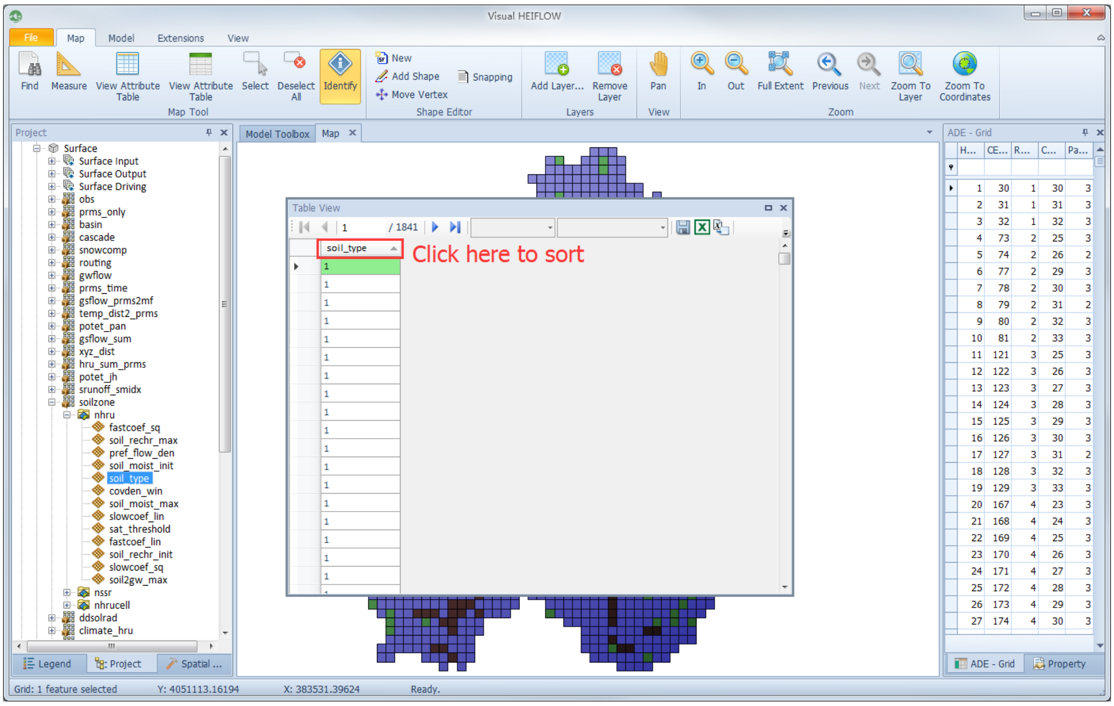

Figure 5-9. Table View of the soil_type parameter

10) You can check parameter values by sorting and plotting. Left click on the column title "soil_type" in the table will sort the values as ascend or descend

order.

11) Right click the column title, and select Plot from the context menu (Figure 5-10):

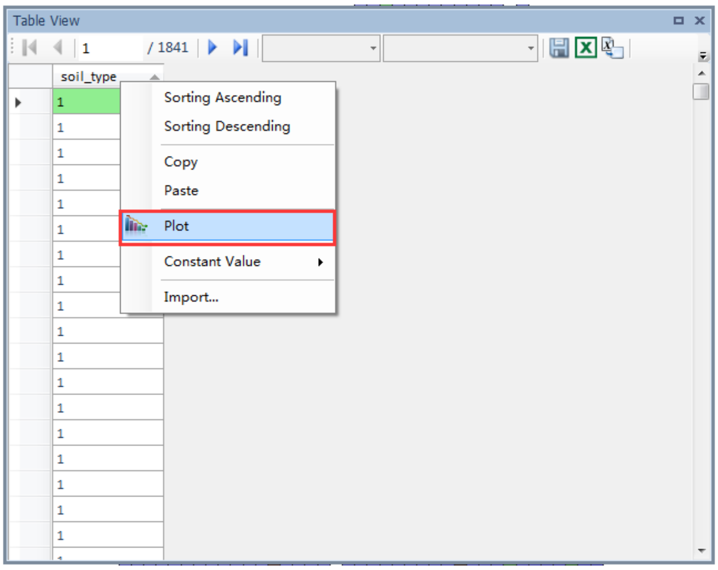

Figure 5-10. Plot parameter values

12) A window entitled “Figure” will be prompted as shown in Figure 5-11.

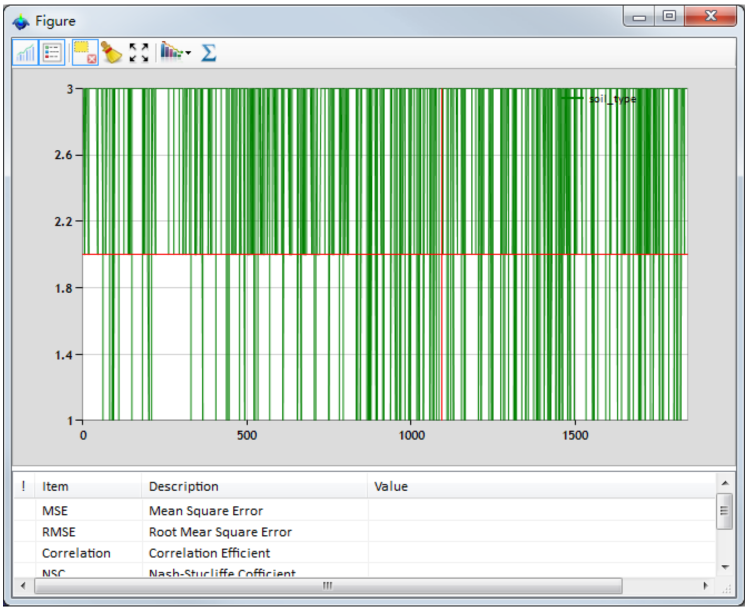

Figure 5-11. Figure shows distribution of the soil_type parameter.

13) Following step 2, create a new coverage named “intcp_cov”, select “landcover” from the Map Layer dropdown list, select intcp from the Packages dropdown list, and then click the button Create Lookup Table (Figure 5-12).

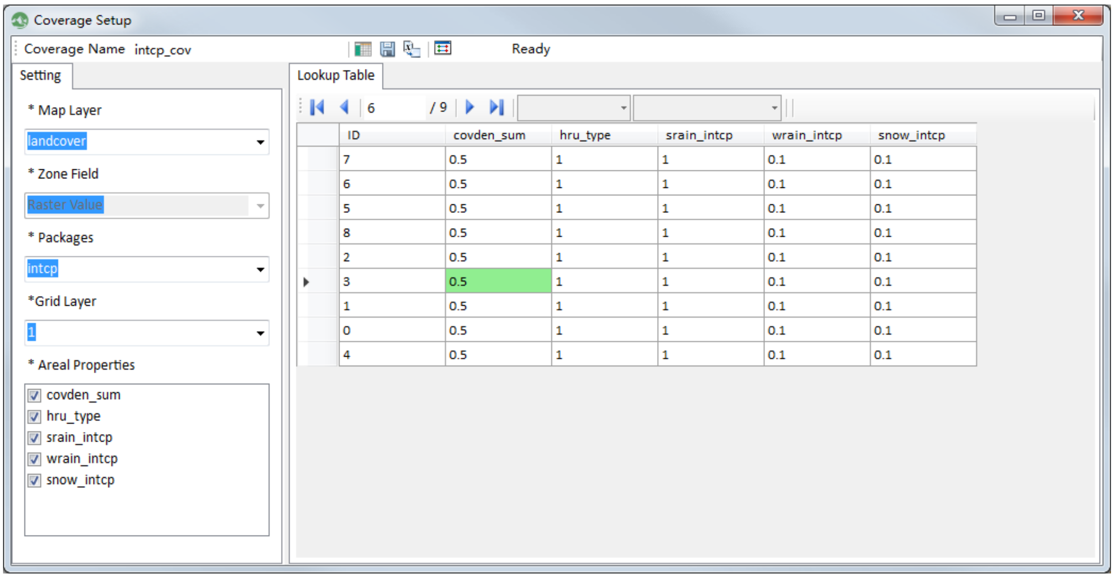

Figure 5-12. Performing intcp package parameterization

14) Import the csv file “landcover.csv” located in the Process Folder, then click Save button and Run Parameterization button.

15) Following step 2, create a new coverage named “snowcomp_cov”, select “landcover” from the Map Layer dropdown list, select snowcomp from the Packages dropdown list, and then click the button Create Lookup Table

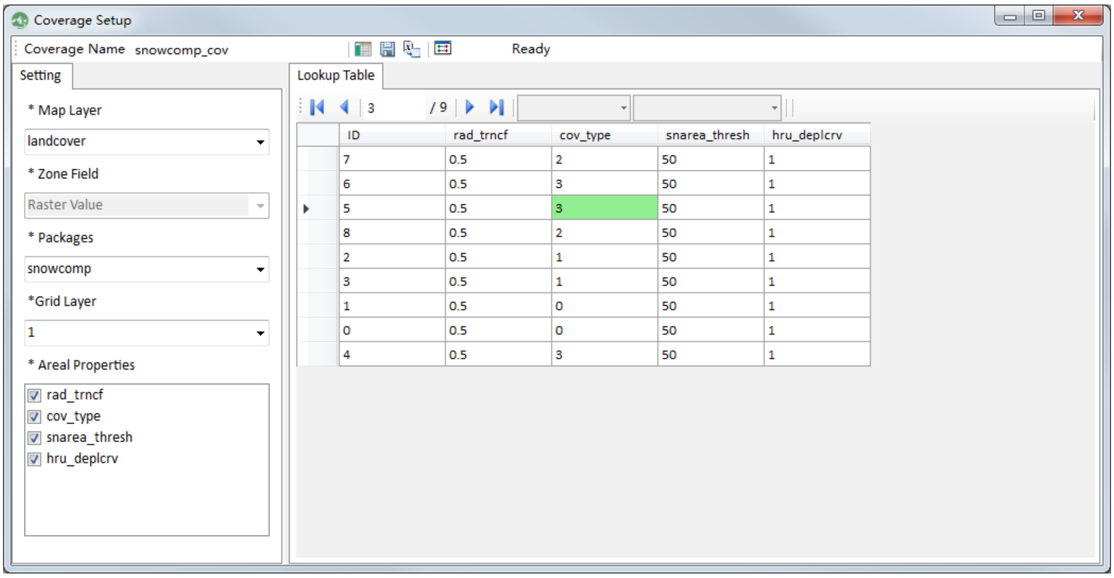

Figure 5-13. Performing snowcomp package parameterization

16) Import the csv file “landcover.csv” located in the Process Folder, then click Save button and Run Parameterization button.

17) Navigate to cov_type parameter, show its map view (Figure 5-14):

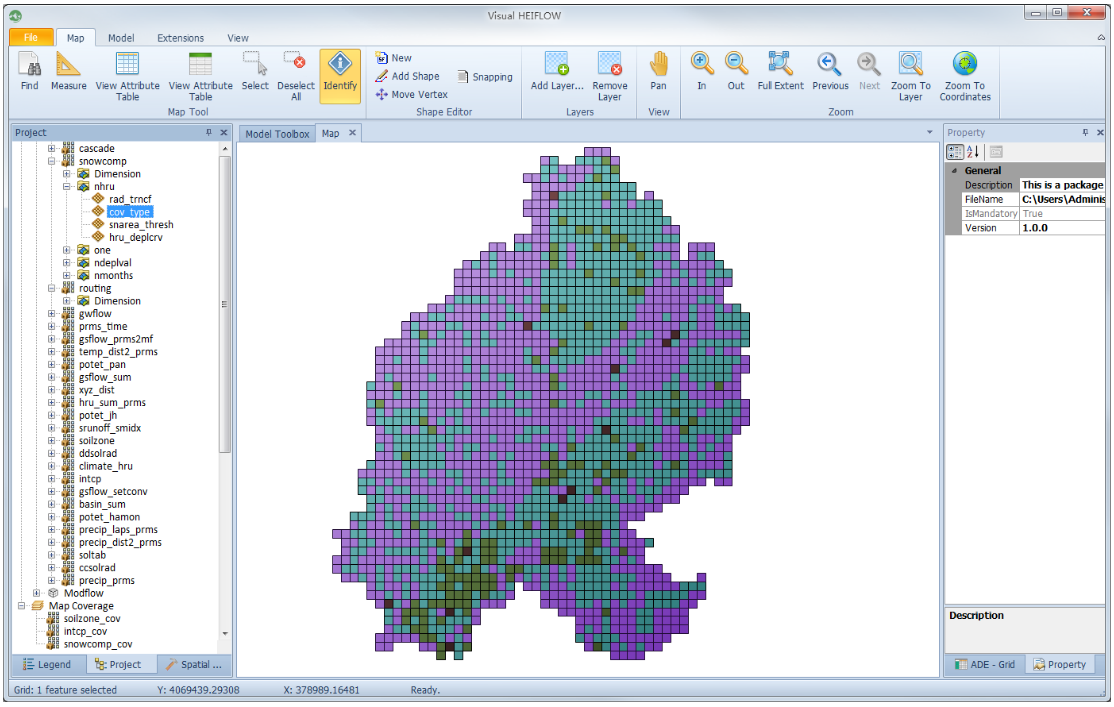

Figure 5-14. Spatial distribution of the cov_type parameter

18) Select File -> Save Project.

5.2 Subsurface model construction

The subsurface model of HEIFLOW is based on MODFLOW. The default MODFLOW packages contained in a HEIFLOW model simulation include BAS6, DIS, LPF, UZF, PCG, OC and SFR. Input files these packages have been generated during the domain decomposition except SFR. These packages and their variables or parameters are displayed in the Project Explorer. Note that the SFR is not ready yet and is indicated by the icon . This section illustrates how to assign values to the MODFLFOW packages through conceptual model approach. A conceptual model is represented either by a shapefile or by a raster. You can develop the conceptual model by using VHF or by using other GIS software like ArcMap.

5.2.1 LPF Package

LPF package contains properties controlling flows between cells at each grid layer. This tutorial provides a conceptual model in terms of raster to represent the hydrogeology zones.

The following describes the procedures of setting these properties layer-by-layer through the conceptual model approach.

1) Select Map -> Add Layer in the Main Menu bar, add “hygeo_zone.tif” placed in the Spatial Data folder into the map;
2) In the Project panel, right click on the Map Coverage, and select New Coverage… from the prompted context menu:

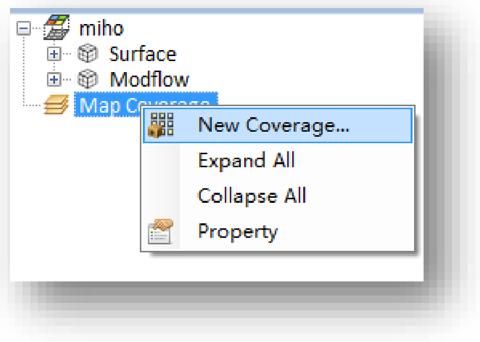

Figure 5-15. Create New Coverage for LPF.

3) In the prompted window entitled “Coverage Setup”, input “lpf_layer1” in the textbox with the name Coverage Name, select “hygeo_zone” from the Map Layer dropdown list, select LPF from the Packages dropdown list, and select 1 from the Grid Layer lpf_layer1.csv (Figure 5-16). Click “Create Lookup Table” in the toolbar, then a lookup will be created and displayed.

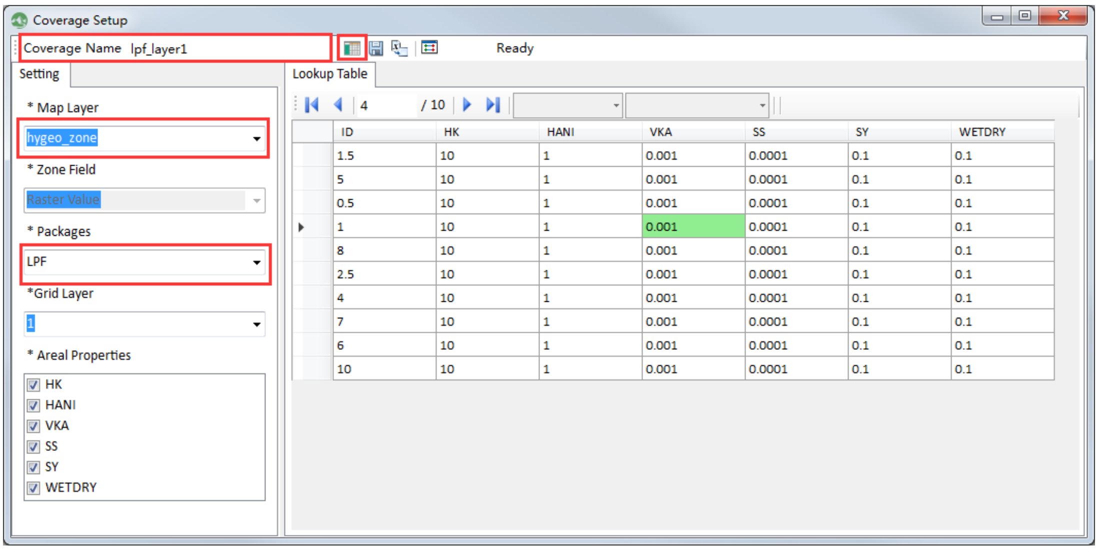

Figure 5-16. Lookup table for the LPF package

4) Click and select the “lpf_layer1.csv” file placed in the Process file folder in the prompted dialog.

5) Click to run parameterization. When finished, you will see the message “successful” shown in the toolbar, then close the Coverage Setup window.

6) In the Project Explorer, locate to the item “HK Layer1” and show its Map View (Figure 5-17).

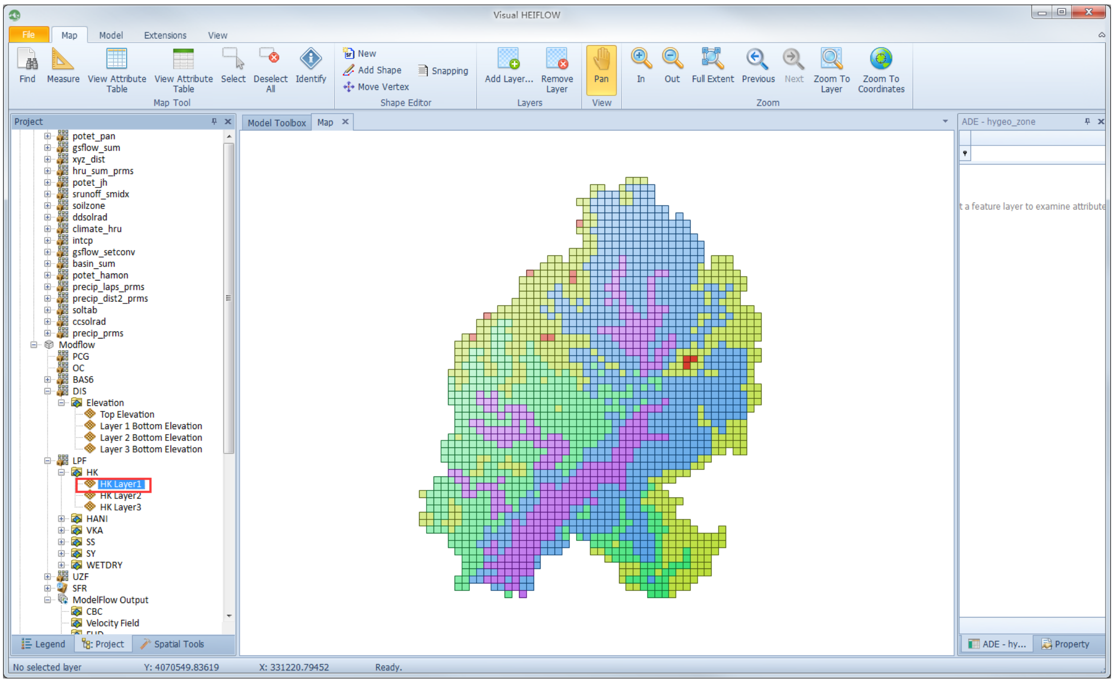

Figure 5-17. Spatial distribution of horizontal hydraulic conductivity (HK) of the first grid layer.

7) Up to now, the flows properties have been set for the first grid layer. Following the steps from 1 to 6, you can set the properties for layer 2 and layer 3. Note that when assigning values to properties of layer 2, you should select “2” from the Grid Layer dropdown list in Figure 5-15 and import the “lpf_layer2.csv” to update the lookup table.

8) In the Project Explorer, right click on the item LPF and select Save from the context menu. Then all changes made to LPF will be saved to the LPF file. VHF allows to save package input file separately.

5.2.2 DIS Package

The DIS package input file contains elevation data of each grid layer. The top elevation of layer 1 has been generated during spatial dissertation, and bottom elevations of layer 1,2 and 3 also have been generated by using constant layer heights, which are defined in Figure 4-11. However, layer heights should be spatially varied. This tutorial provides three raster files that represent the layer heights, including "layer1_height.tif", "layer2_height.tif" and "layer3_height.tif". These raster files are used to define bottom elevations through follow steps:

1) Select Map -> Add Layer…, and add "layer1_height.tif", "layer2_height.tif" and "layer3_height.tif" into the Map.

2) Select View -> Model Toolbox, select the Tools panel in the Model Toolbox , and locate to the tool "Extract from Raster " (Figure 5-18).

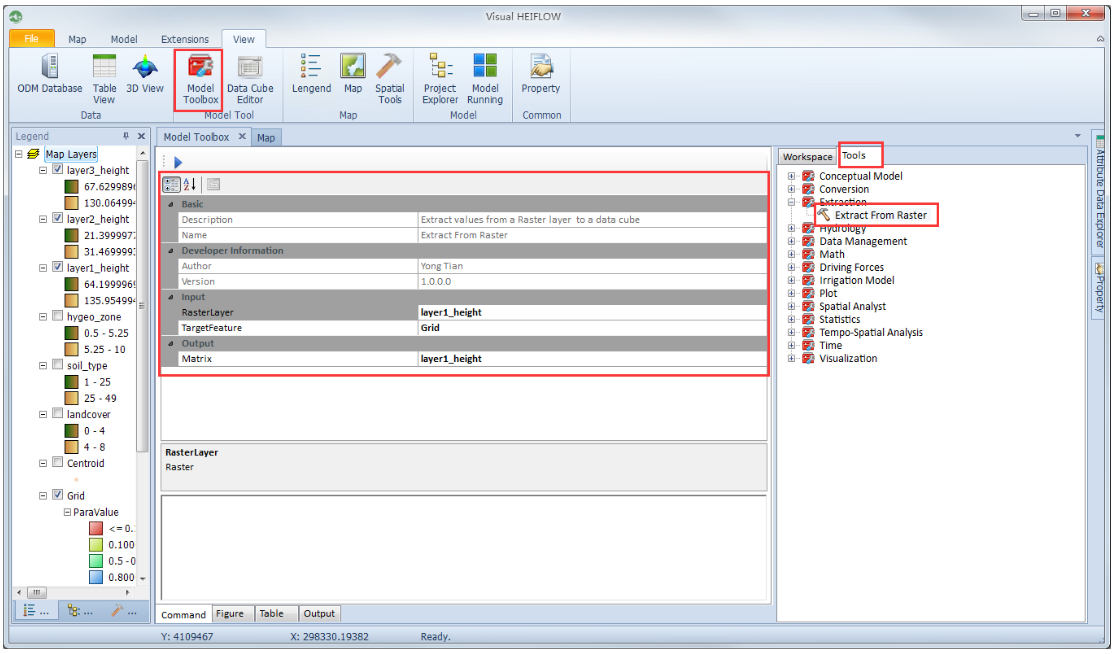

Figure 5-18. Model Toolbox interface

3) Double-click on the "Extract from Raster ", the grid used to specify input and output of the tool is displayed. Select "layer1_height" from the RasterLayer dropdown list, and select "Grid" from the TargetFeature dropdown list. Input "height1" in the Matrix textbox.

4) Click ▶ to run the tool. A dialog entitled "Tool Progress" will appear to

show the running progress (Figure 5-19). When running finished, click Close in the dialog.

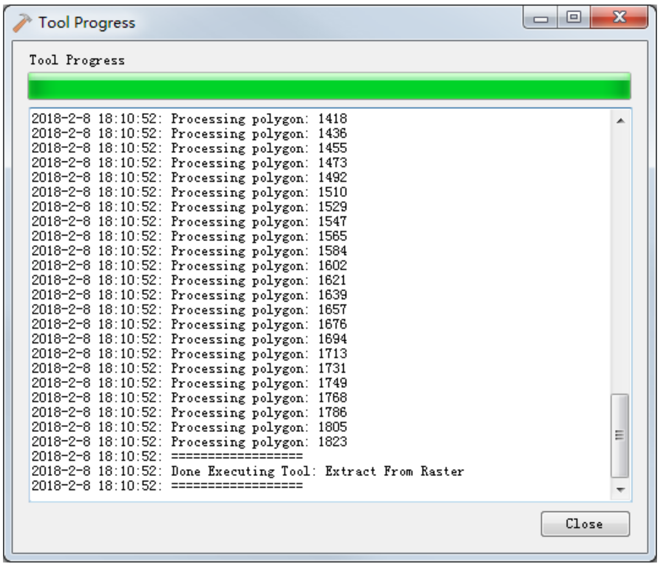

Figure 5-19. Tool Progress dialog

5) Select Workspace panel located in the right side of the Model Toolbox view, a data cube named “height1” is added (Figure 5-20).

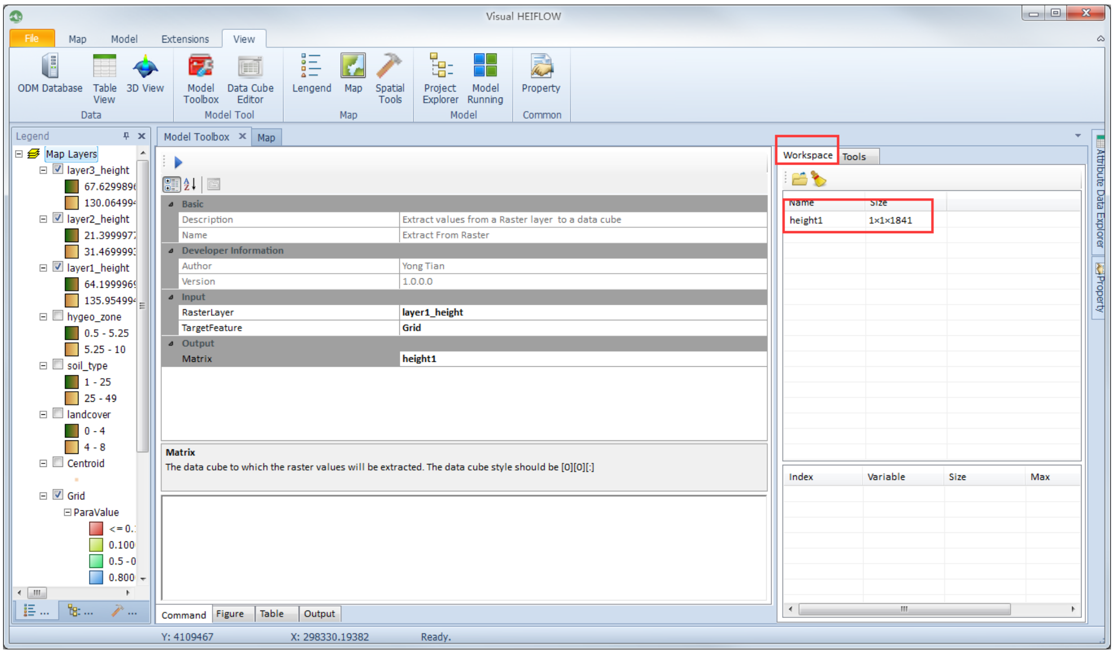

Figure 5-20. Tool Progress dialog

6) In the Project Explorer, locate to the item “Top Elevation”, right click on it and select “Add to toolbox…”. Then the data cube named “Elevations” will appear in the Workspace (Figure 5-21).

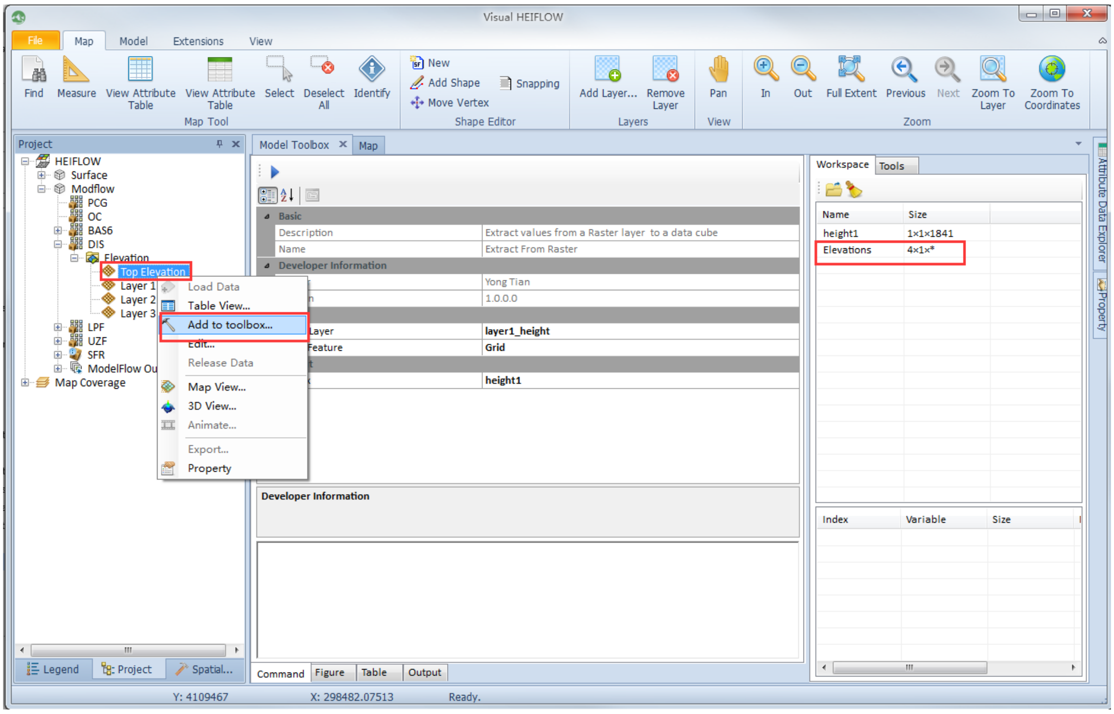

Figure 5-21. Adding the data cube variable “Elevations” to the workspace.

7) Locate the “Minus” tool in the Tools Panel, and double click it. Set its Input and Output according to Figure 5-22:

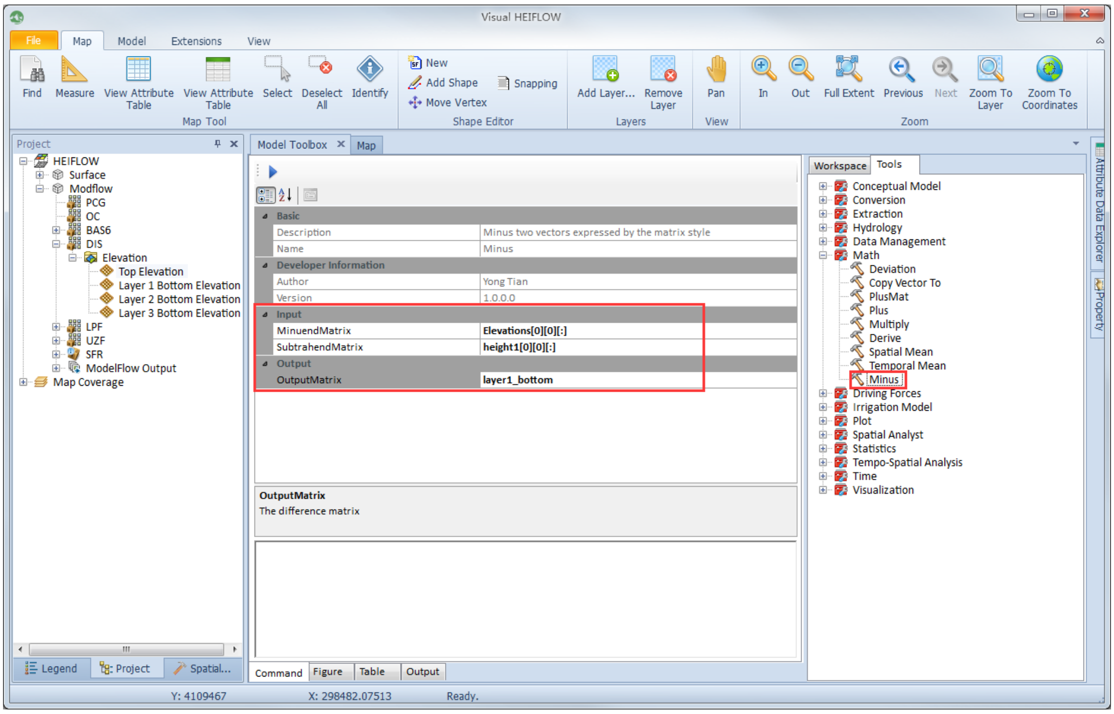

Figure 5-22. Calculating bottom elevation of layer 1 by using the Minus tool.

8) Click Run, then a data cube named “layer1_bottom” appears in the Workspace.

9) Double click the "Copy Vector To" tool, set Input as shown in Figure 5-23:

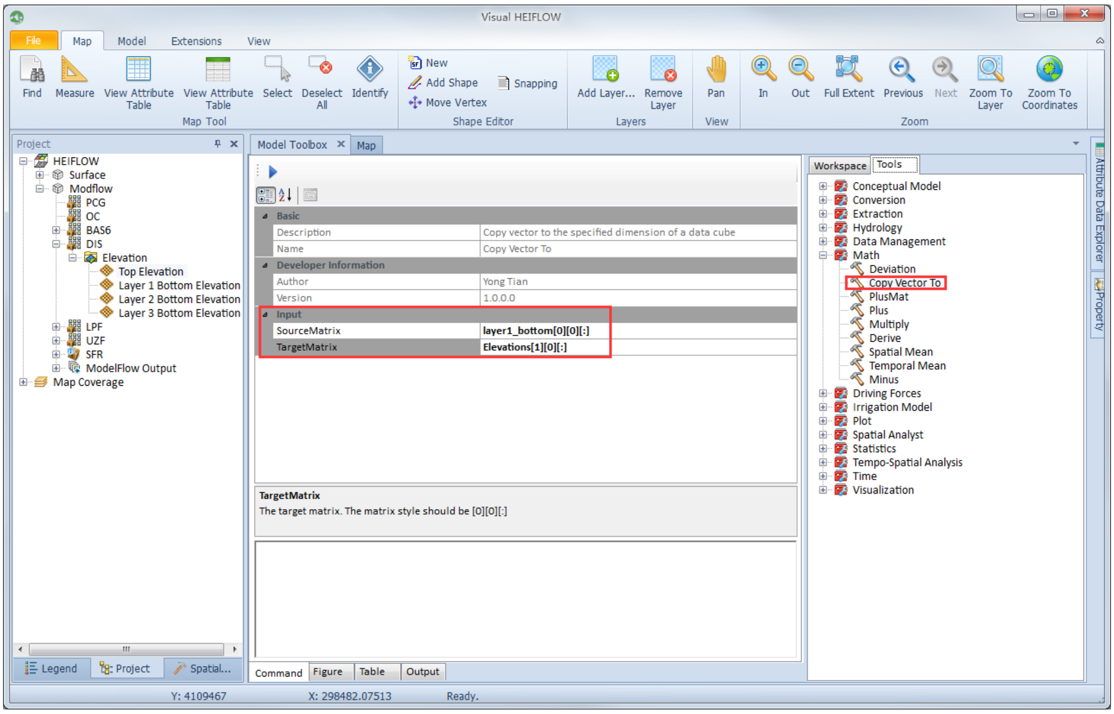

Figure 5-23. Copy values to the Elevations data cube

10) Click Run. After finished, right click on the "Layer 1 Bottom Elevation" in the Project Explorer, and select Map View to see its spatial distribution (Figure 5-24)

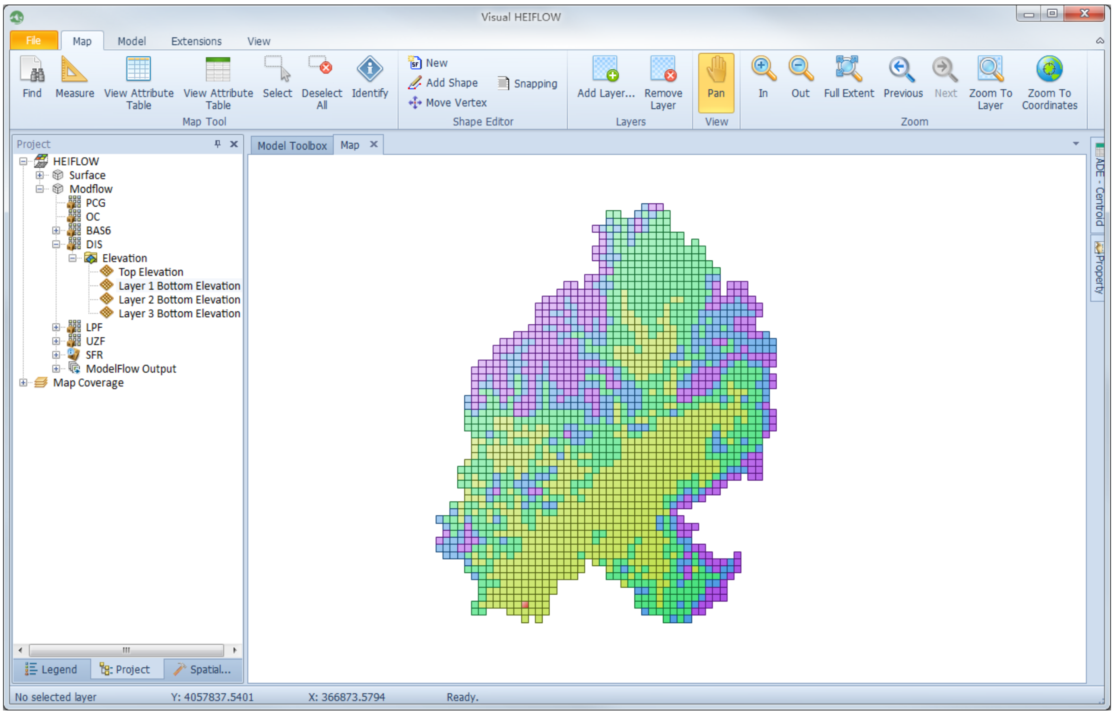

Figure 5-24. Spatial distribution of Layer 1 Bottom Elevation.

11) Repeat the steps 1-5, extract height of layer 2 from the raster file “layer2_height.tif” to the data cube “height2”, and extract height of layer 3 from the raster file “layer3_height.tif” to the data cube “height3”.

12) Double click the “Minus” tool. Set its Input and Output according to Figure 5-25, and click Run. Note that MinuendMatrix should be “Elevations[1][0][:]” and SubtrahendMatrix should be height2[0][0][:].

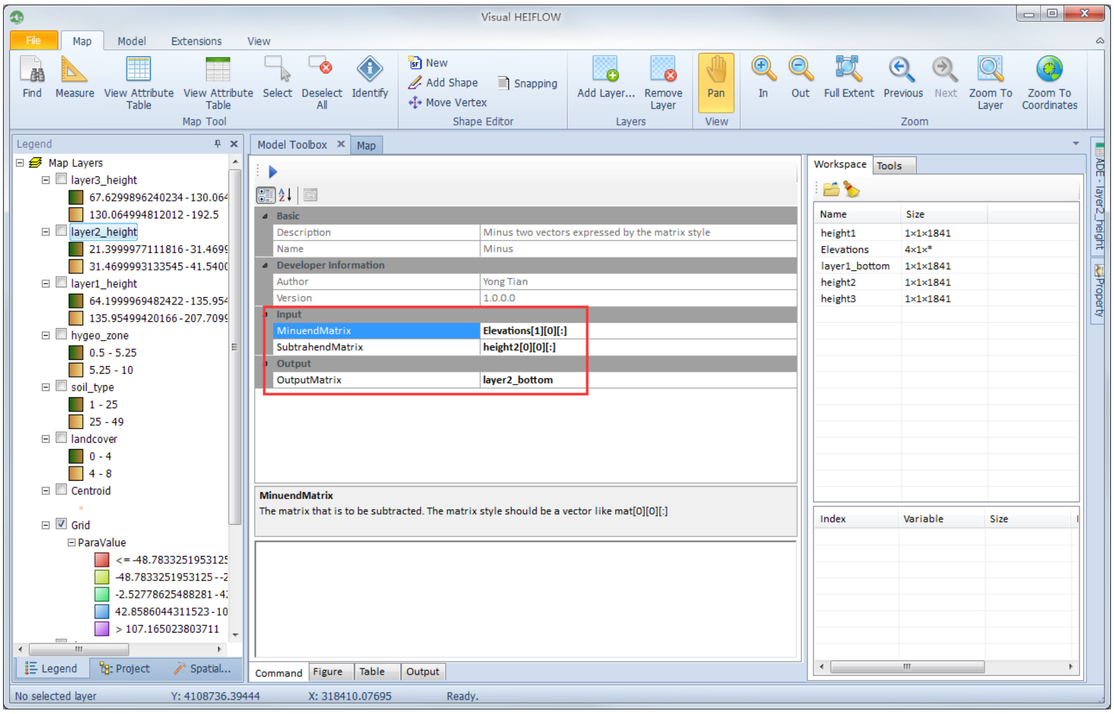

Figure 5-25. Calculating bottom elevation of layer 2 by using the Minus tool.

13) Double click the “Copy Vector To” tool, set Input as shown in Figure 5-26, and click Run.

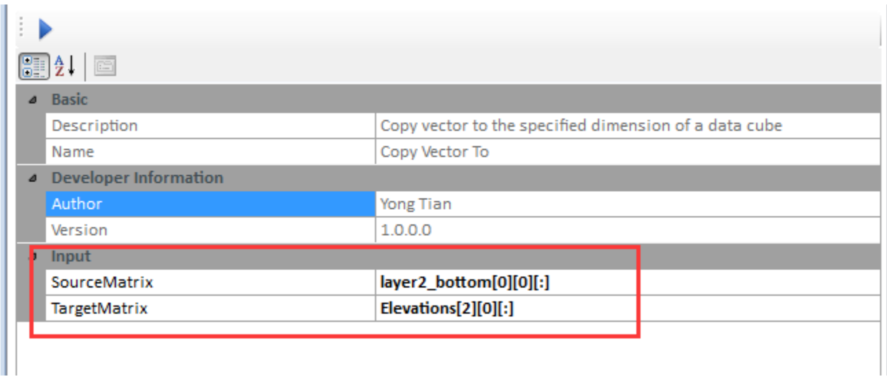

Figure 5-26. Copy values to the Elevations data cube

14) Double click the “Minus” tool. Set MinuendMatrix as “Elevations[2][0][:]”,

set SubtrahendMatrix as “height3[0][0][:]”, and set OutputMatrix as “layer3_bottom”. Then Click Run. Double click the “Copy Vector To” tool, set SourceMatrix as “layer3_bottom[0][0][:]” and TargetMatrix as “Elevations[3][0][:]”, then click Run.

15) In the Project Explorer, right click on the item DIS, and select Save in the prompted context menu.

5.2.3 UZF Package

The UZF1 Package input file contains information about the hydraulic properties of the unsaturated zone, infiltration rate, potential evapotranspiration (PET) and others. HEIFLOW only requires infiltration rate and PET of steady state, which could be estimated based on observed meteorological data. This procedure will be illustrated in Section 6.

5.2.4 SFR Package

Input to the SFR package is very complicated. VHF provides a tool to automatically generate the input.

5) Select Map -> Add Layer…, locate to the current project folder(e.g., C:\Users\Administrator\Documents\MIHO\) and add the “demad8.tif” placed in the Processing sub-folder (Figure 5-27).
6) In the Model Toolbox View, select Conceptual Model -> Streamflow Routing and double click it, set the inputs as shown in Figure 5-28.
7) Click Run. It requires considerable time to finish the calculation. Please be patient.
8) After the calculation finished, a new layer named “SFR” is added into the map (Figure 5-29).
9) In the Project Explorer, right click on the SFR, and select Save.

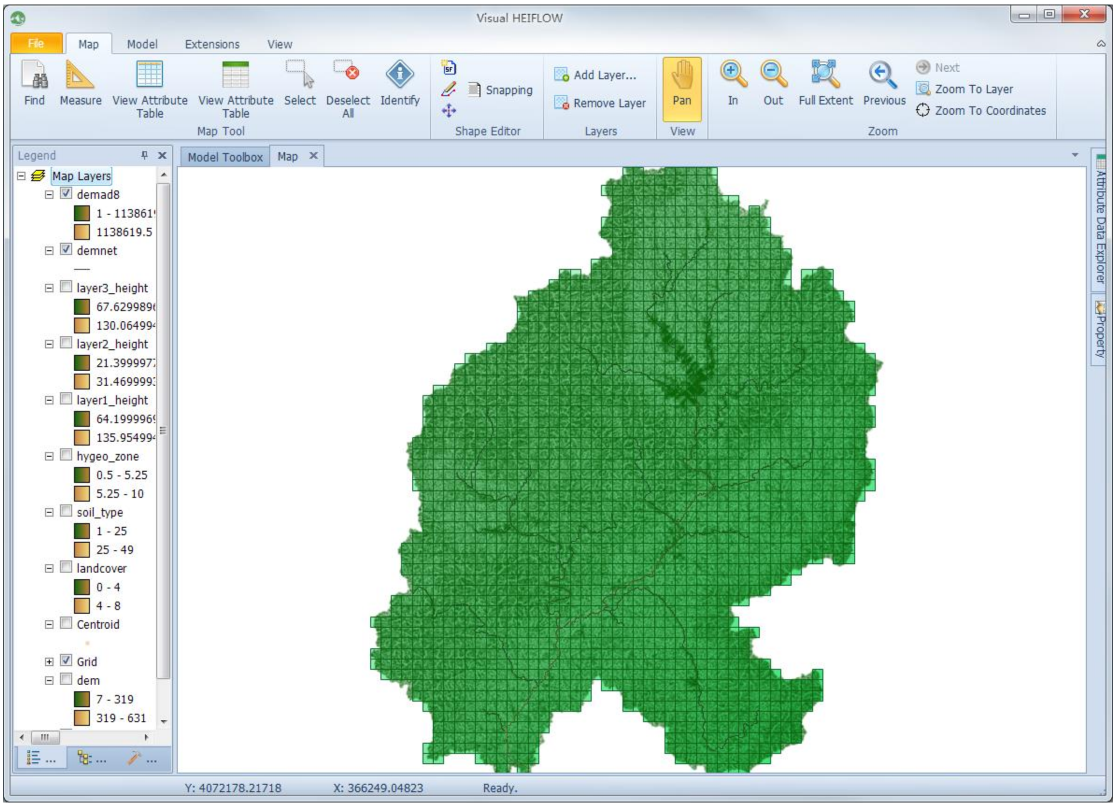

Figure 5-27. Adding “demad8.tif” to the map, this raster represent accumulated drainage area.

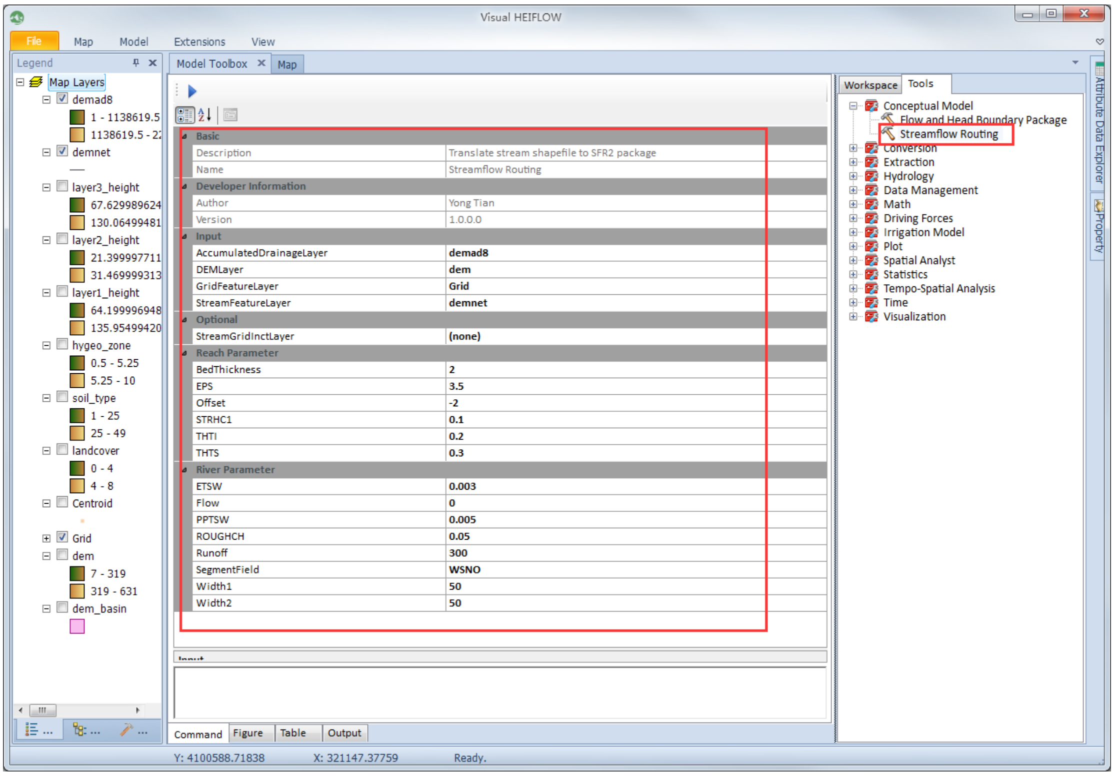

Figure 5-28. Inputs of the Streamflow Routing tool

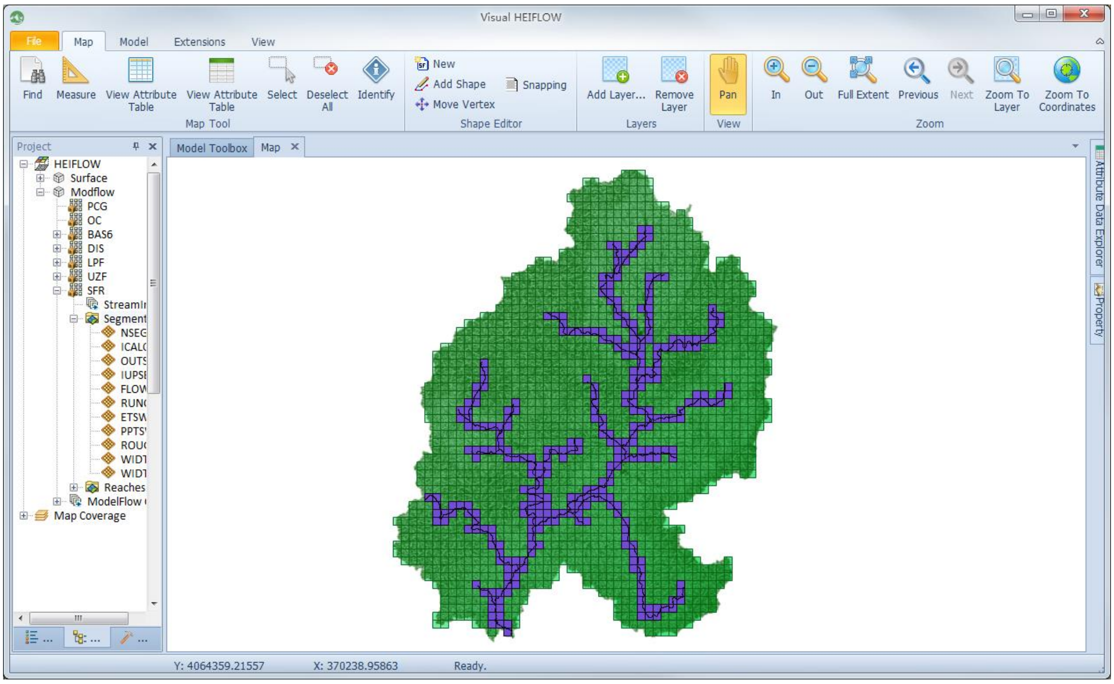

Figure 5-29. Generated SFR grid

5.2.5 FHB Package

The Flow and Head Boundary (FHB) package is used for specified head cells and specified flow cells whose properties can vary within a stress period. VHF provides a tool to generate FHB input file. The procedures of generating the input file are as follows:

1) Select Map -> Add Layer…, locate to the Tutorial\Spatial Data file folder and add the “FHB_source.shp” into the Map.
2) In the Model Toolbox View, select Conceptual Model -> Flow and Head Boundary Package and double click it, set the inputs as shown in Figure 5-30.
3) Click Run. When finished, a layer named “FHB” will be added to the map (Figure 5-31).
4) In the Project Explorer, right click on the FHB item, and select Save.

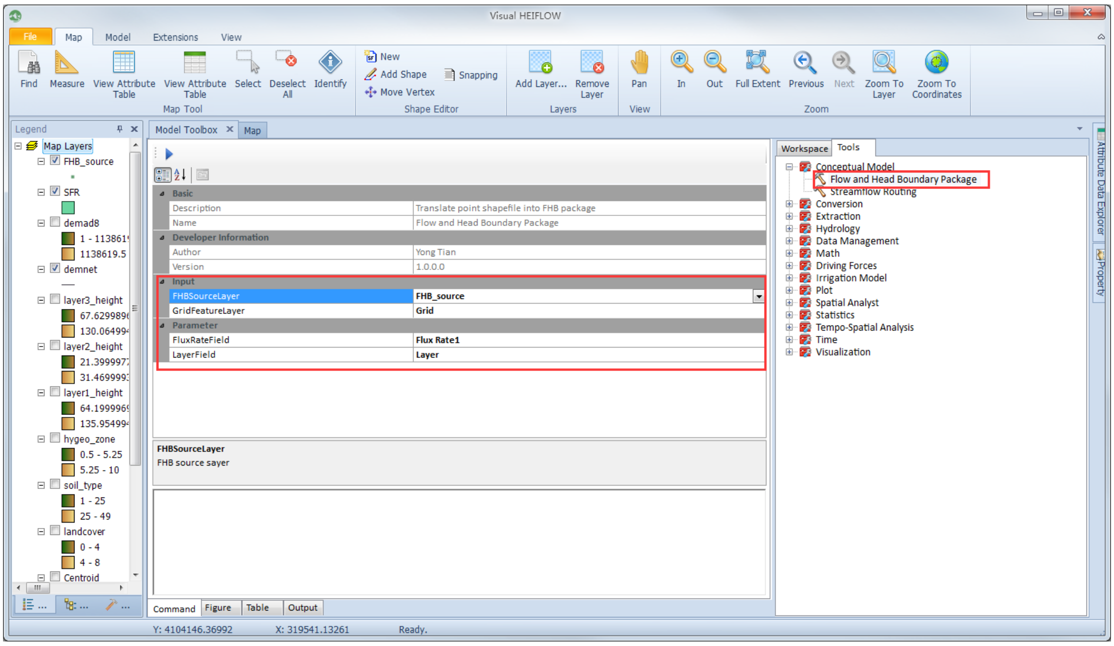

Figure 5-30. Input and parameter settings for the Flow and Head Boundary Package tool.

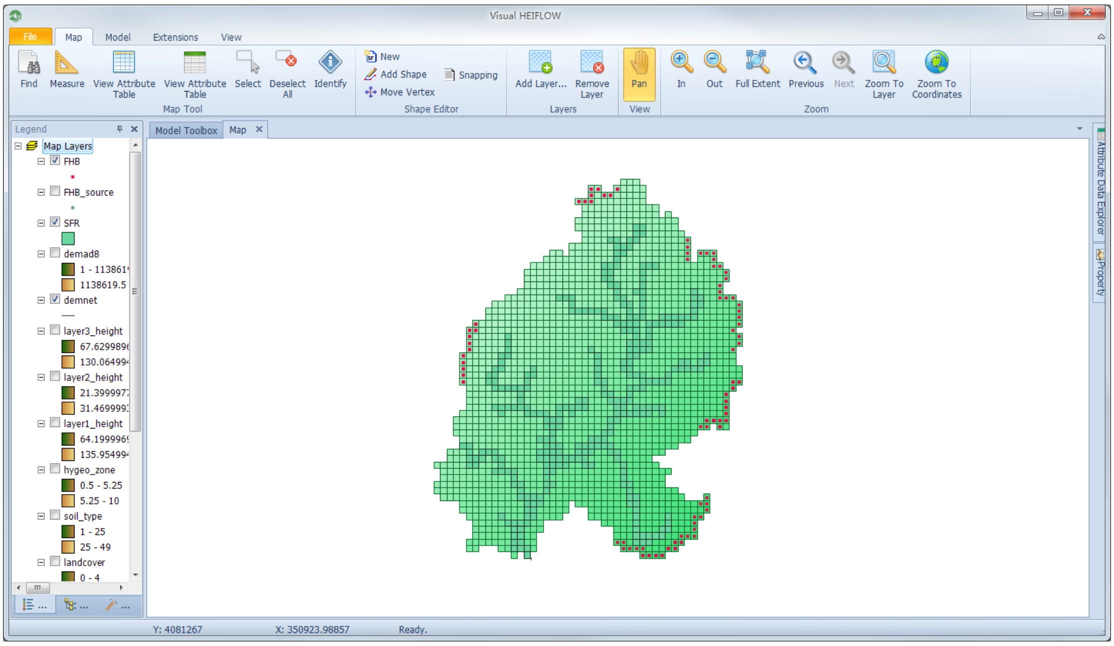

Figure 5-31. Generated FHB layer.

5.2.6 PCG Package

The Preconditioned Conjugate-Gradient (PCG) package is used to solve the finite difference equations in each step of a MODFLOW stress period. Input to PCG defines parameters of the solver. You can directly modify these parameters in the Property Panel as follows:

1) In the Project Explorer, right click on the FHB item, and select Property. The Property Panel appears (Figure 5-32).

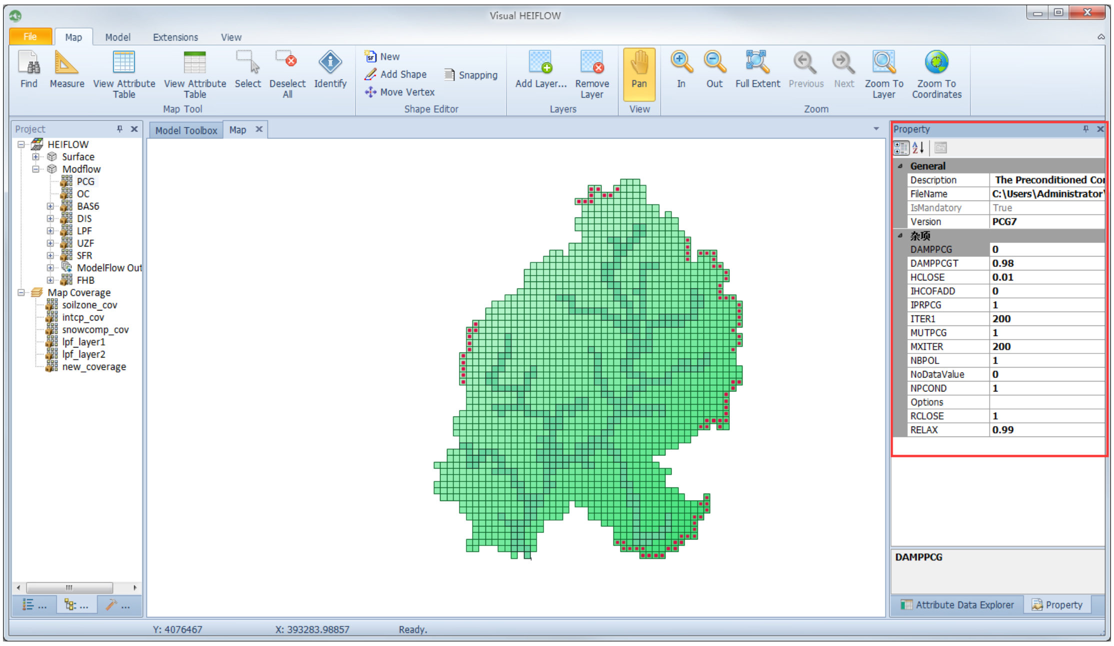

Figure 5-32. Property of the PCG package.

2) You can modify the solver parameters in the Property Panel.

3) If any changes made to the parameters, right click on the FHB item in the Project Explorer, and select Save.

5.3 Calculation of cascade

HEIFLOW uses the concept of cascade to route overland flow and interflow among HRUs, streams and lakes. The calculation of cascade requires information contained in the input files of DIS and SFR packages. With the generation of those input files, VHF provides a tool to calculate the cascade parameters in an automated manner.

1) Select Model -> Cascade in the Main Menu Bar, input IDs of cell, row and column following ;

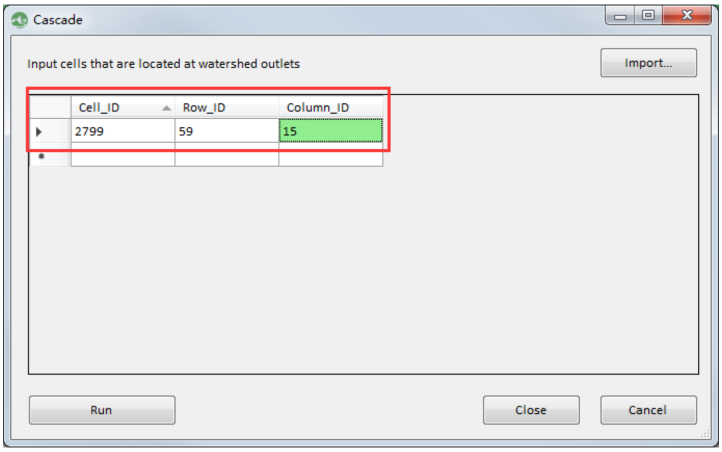

Figure 5-33. ID of the cell that are located at the watershed outlet.

2) Click Run. When finished, click Close.

6 Generation of Driving Force and Calibration Data Sets

6.1 Generation of Driving Force

HEIFLOW requires daily precipitation, daily maximum temperature, daily minimum temperature, and daily potential evapotranspiration as driving forces. In the MIHO catchment, there are 7 rainfall gaging stations and 1 meteorological station. These data are used to generate climate inputs.

At first, templates of the required climate inputs could be generated.

1) Select Model -> Weather Generator in the Main Menu bar, the Weather Generator dialog will be prompted.

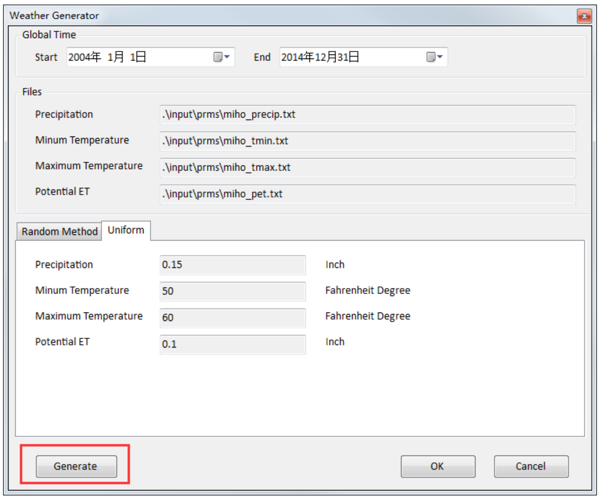

Figure 6-1 Weather Generator dialog.

2) Click Generate button. All the template climate inputs will be generated and saved to the Input\PRMS folder. Then click OK.

Next,

Observed precipitations are interpolated to each grid cell from the 7 rainfall stations using Inverse Distance Weighting (IDW) method.

1) Select Map -> Add Layer in the Main Menu bar, add “rainfall_sites” placed in the Tutorial\Spatial Data folder into the map;

2) In the Model Toolbox View, select Spatial Analyst -> Inverse Distance Weighting and double click it, set the inputs as shown in.

3) Under the Input category, left click in the DataFileName, a browse button will appear (Figure 6-2). Click the browse button, in the prompted dialog select the “rainfall7.csv” file placed in the Tutorial\Meteorological data folder.

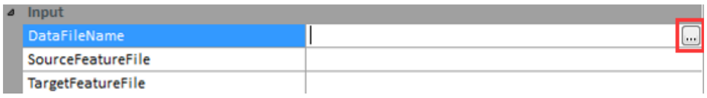

Figure 6-2. Browse button used to select data file.

4) Set the other Input and Parameter as shown in Figure 6-3. Click Run. When finished, a matrix named “ppt” appears in the Workspace.

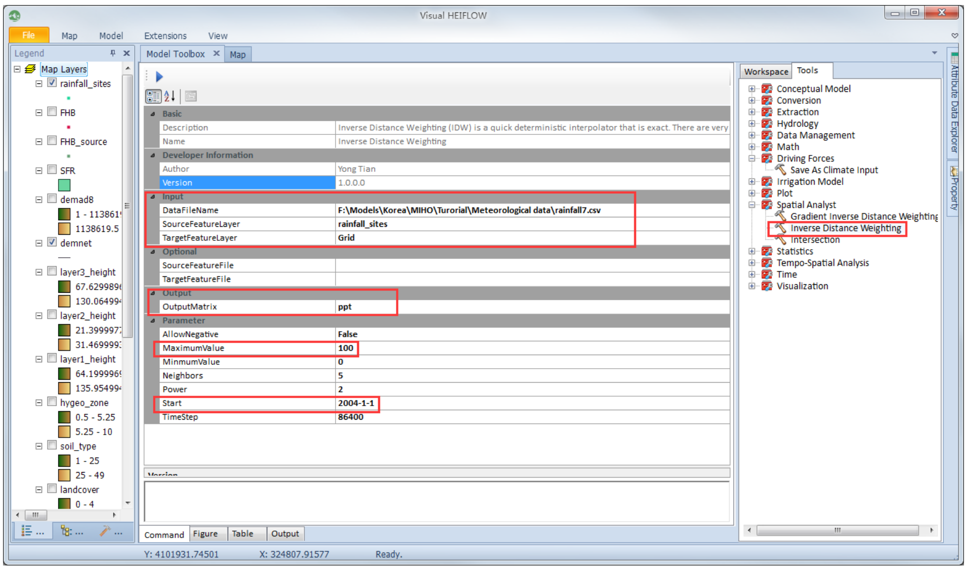

Figure 6-3. Input and Parameter settings for the Inverse Distance Weighting tool.

5) In the Model Toolbox View, double click the tool Driving Forces -> Save As Climate Input, set its input and output as shown in Figure 6-4

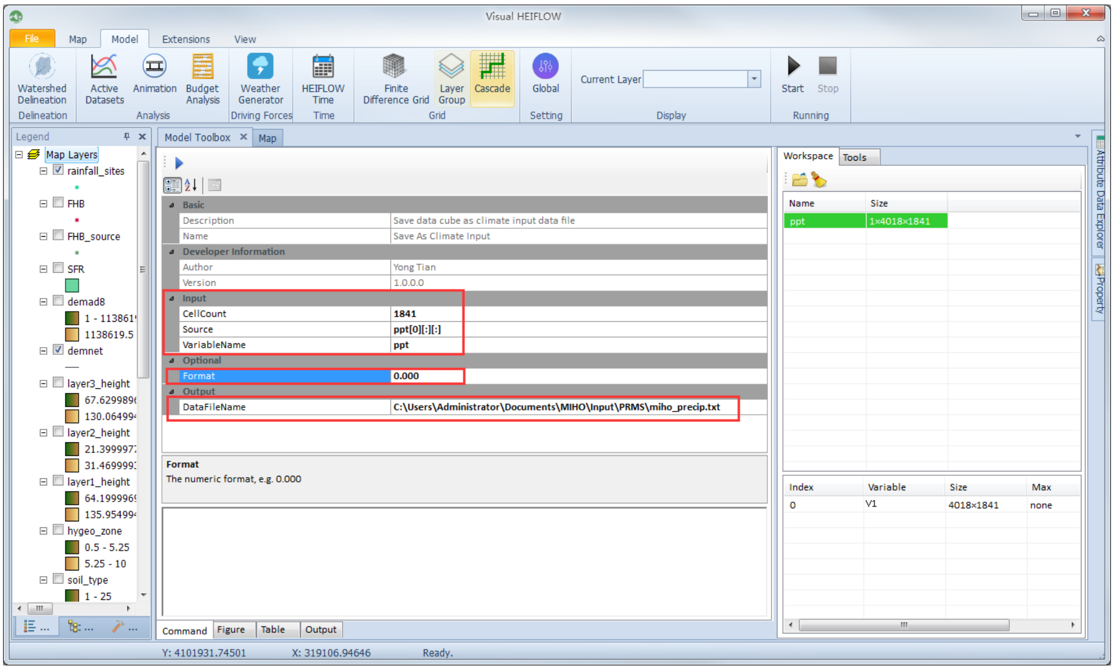

Figure 6-4. Input and output of the Save As Climate Input tool.

6) In the Model Toolbox View, double click the tool Conversion -> From Multiple Variable Text File, set its input and output as shown in Figure 6-5.

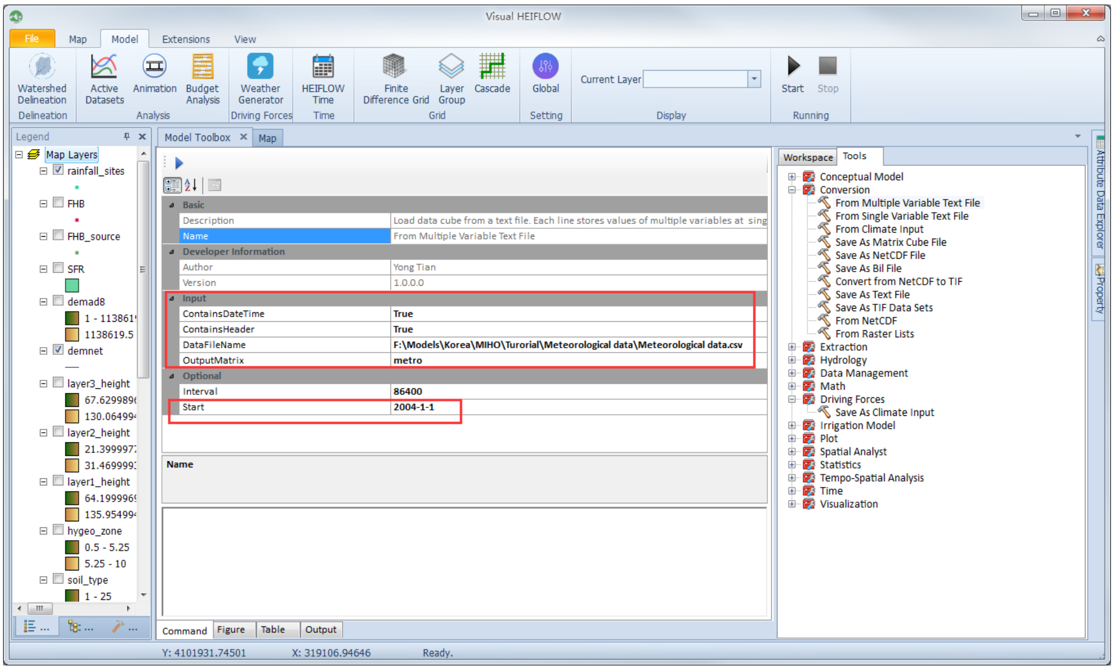

Figure 6-5. Input and output of the From Multiple Variable Text File tool.

7) Click Run. When finished, a matrix named "metro" will appear in the

Workspace.

8) In the Model Toolbox View, double click the tool Driving Forces -> Save As Climate Input, set the input and output as shown Figure 6-6 to save PET input file.

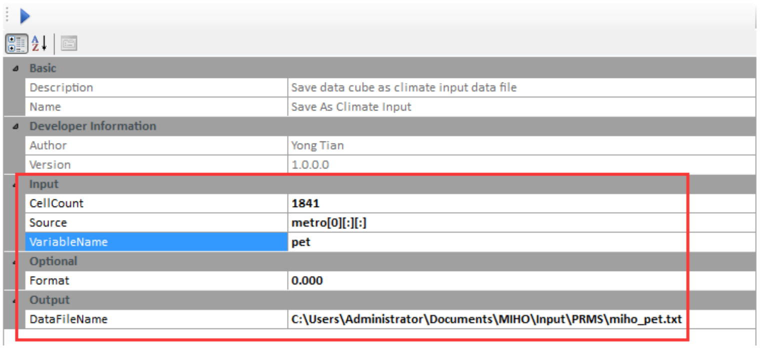

Figure 6-6. Settings used to generate PET input file.

9) Change input and output of the Save As Climate Input tool according to Figure 6-7 to save Maximum Temperature input file.

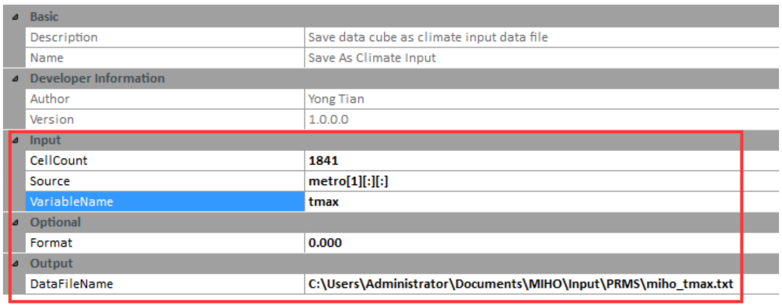

Figure 6-7 Settings used to generate Maximum Temperature input file.

10) Change input and output of the Save As Climate Input tool according to Figure 6-8to save Minimum Temperature input file.

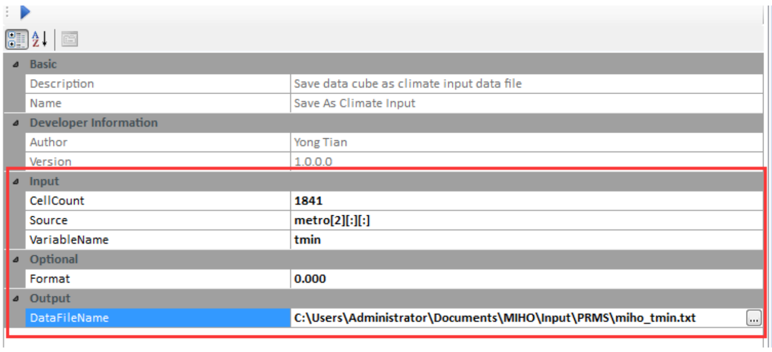

Figure 6-8 Settings used to generate Minimum Temperature input file.

6.2 Generation of Calibration Data Sets

Streamflow and groundwater observed at point stations are usually used to calibrate hydrological data. In VHF, these point observation time series are stored in a relational database (Microsoft Access 2013 or higher). The storage follows a standard called Observational Database Model (ODM). This tutorial provides a sample database, which contains daily streamflow observed at a hydrological gaging station, and daily groundwater level observed at 17 groundwater monitoring wells for the Miho catchment. VHF also provides a tool that enables users to import external data into the ODM database.

1) Select View -> ODM Database, a docking panel entitled “Database” appears at the right side of the main window (Figure 6-9).

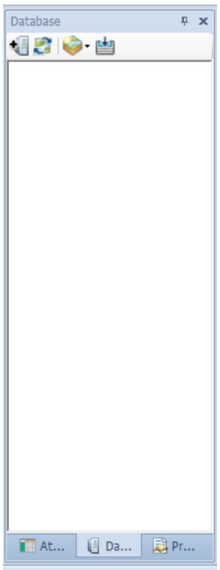

Figure 6-9. Docking panel of the Database Manager.

2) Click the Open an ODM database button, locate to the Tutorial\Database folder and select the “MIHO.mdb” file in the prompted dialog. After the databased loaded, all stations and observed time series made at these stations are presented in the Database Manager in a tree-view (Figure 6-10).
3) Click Import data to database button, then a dialog entitled “ODM Database Manager” is prompted. This dialog allows users to import data into the

dataset.

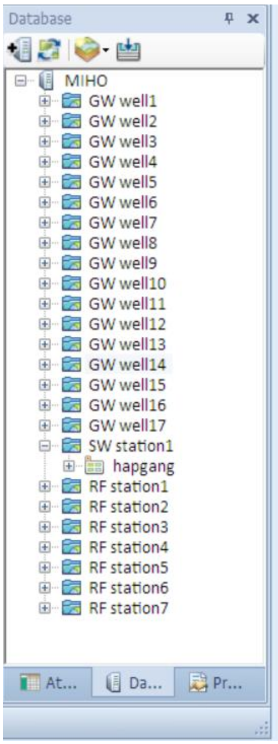

Figure 6-10. The loaded MIHO database.

4) As we known, a relational database uses tables to store and organize data. The main tables contained in the ODM database include: Sites, Variables, DataValues, DataTypeCV, etc. Users can import csv file or Excel file into a specified table. Select a table that you want to import from the table name list, then data content of the selected table will be displayed (Figure 6-11).

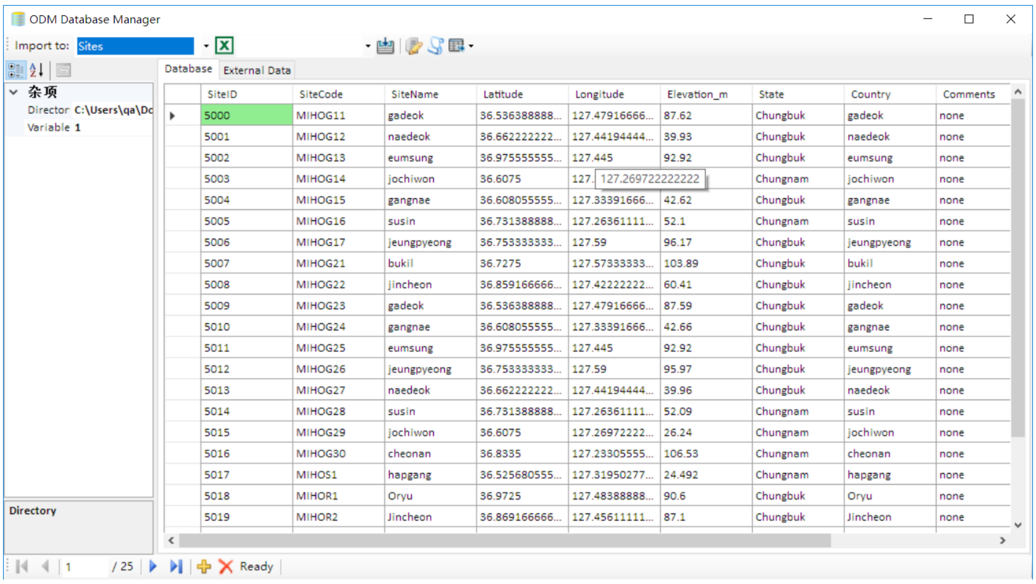

Figure 6-11. The Sites table.

5) You can export the table to a csv file by clicking the Default Export. You can also import data from an existing Excel or csv file. The layout of the imported file must follow that of the corresponding table.

7 Running a Simulation

All the HEIFLOW input files have been generated up to now. You can start a simulation as follows:

1) Select Model -> Start in the Main Menu Bar.

8 Analyzing Simulation Results

8.1 Analysis of streamflow results

At first, we perform comparison of simulated and observed streamflow as follow:

1) In the Project Explorer, right click on the item SFR Output under the Modflow Output, and select Advanced…. A dialog entitle "SFR Package" is prompted to you (Figure 8-1).

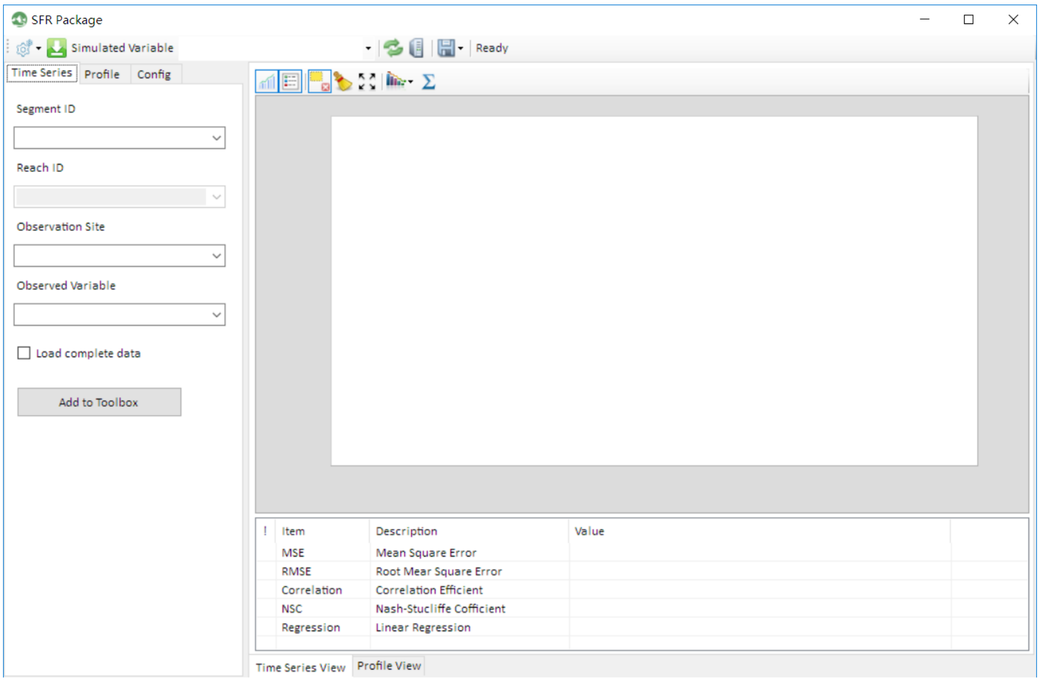

Figure 8-1. The output analysis dialog for SFR package.

2) Click the button, then all the variable related to stream simulation at the last reach of each segment will be loaded.

3) Select "Flow out of stream" in the Simulate Variables combox. Select "35" from the combox below the Segment ID. Then daily streamflow simulated at the outlet of Miho catchment is shown using line curve (Figure 8-2).

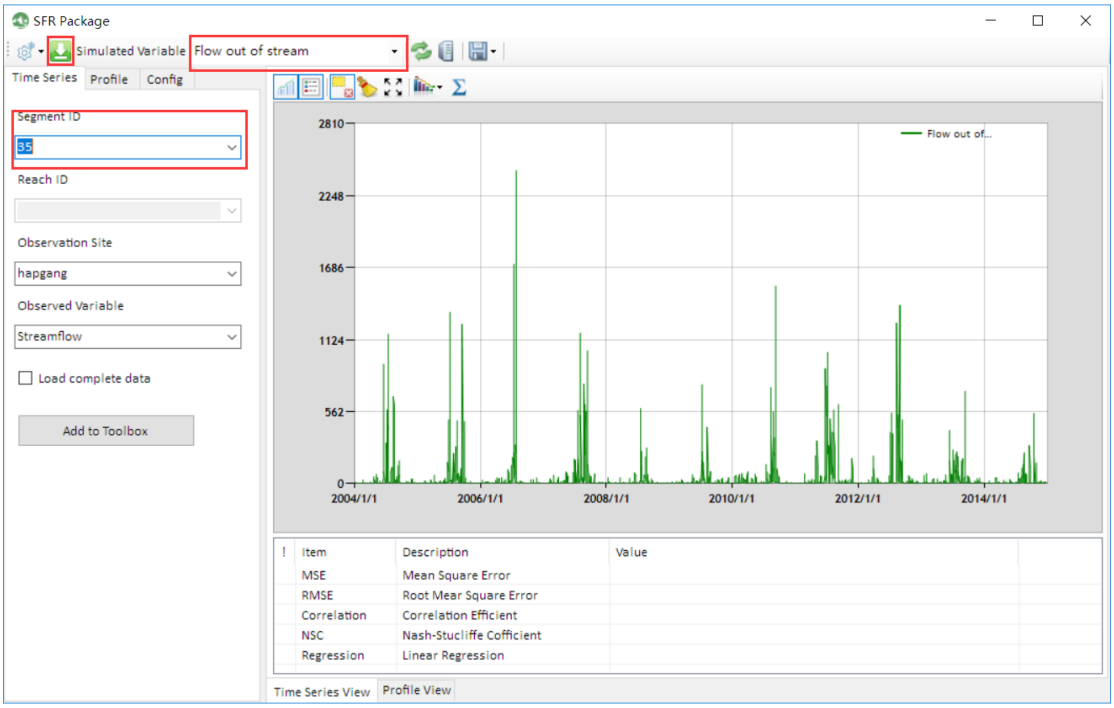

Figure 8-2. Simulate daily streamflow at the outlet of the Miho catchment.

4) Click the button to refresh database source, then all stations stored in the ODM database will be listed in combox below the Observation Sites.

5) Uncheck (Clear existed series) in the figure panel. Select "hapgang" from the Observation Site dropdown list. Then the observed streamflow will also be presented in the figure (Figure 8-3).

6) Click in the figure panel, all the existed curves in the figure will be cleared. Change to the Config panel, select Month in TimeUnits.

7) Select "35" from the Segment ID dropdown list, and select "hapgang" from the Observation Site dropdown list. This will compare simulated and

observed streamflow at monthly time scale (Figure 8-4).

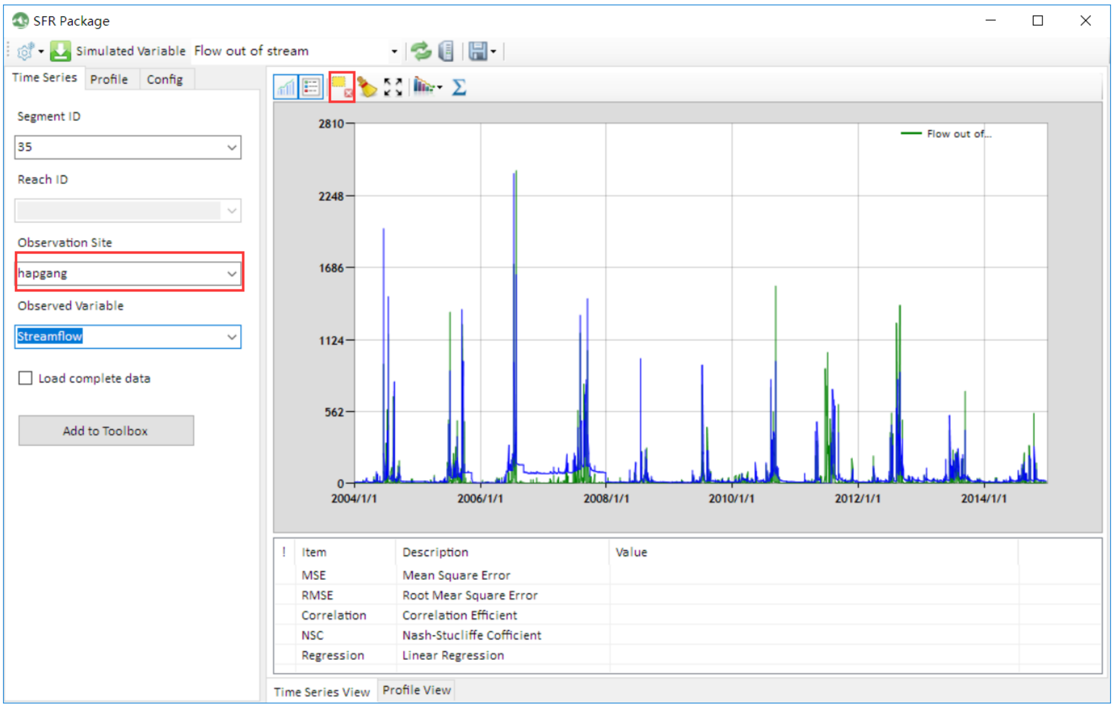

Figure 8-3. Comparison of simulated and observed daily streamflow at the Hapgang gauging station.

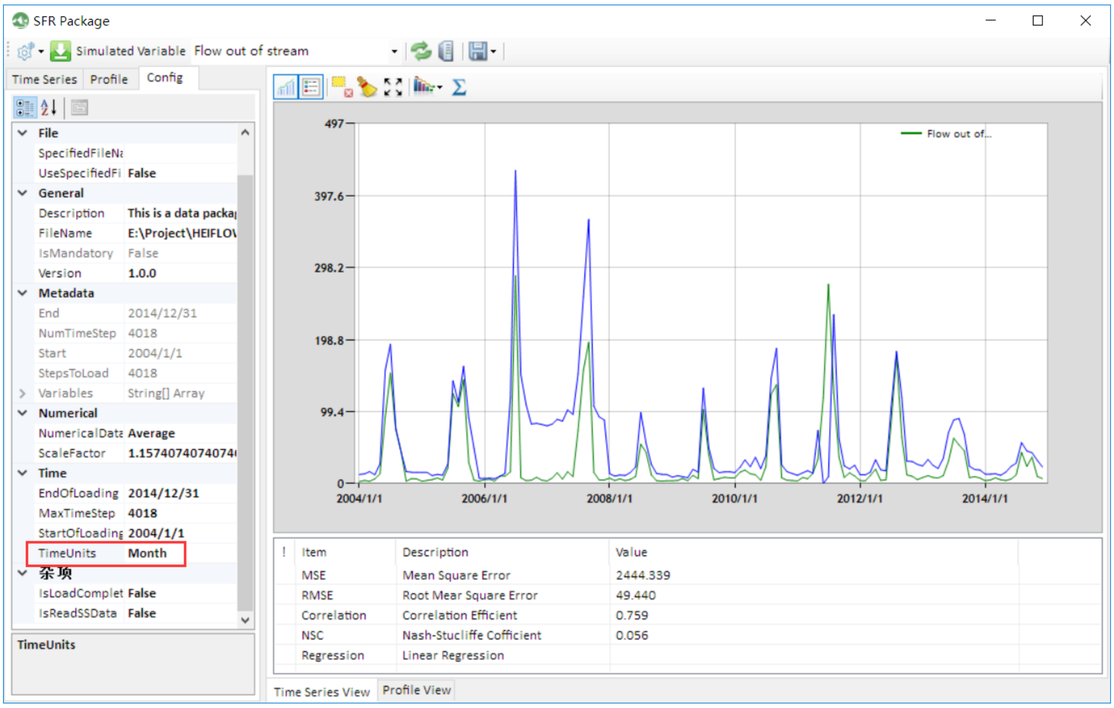

Figure 8-4. Comparison of simulated and observed monthly streamflow at the Hapgang gauging station.

8) Click Σ to calculate statistics between simulated and observed time series (Figure 8-5).

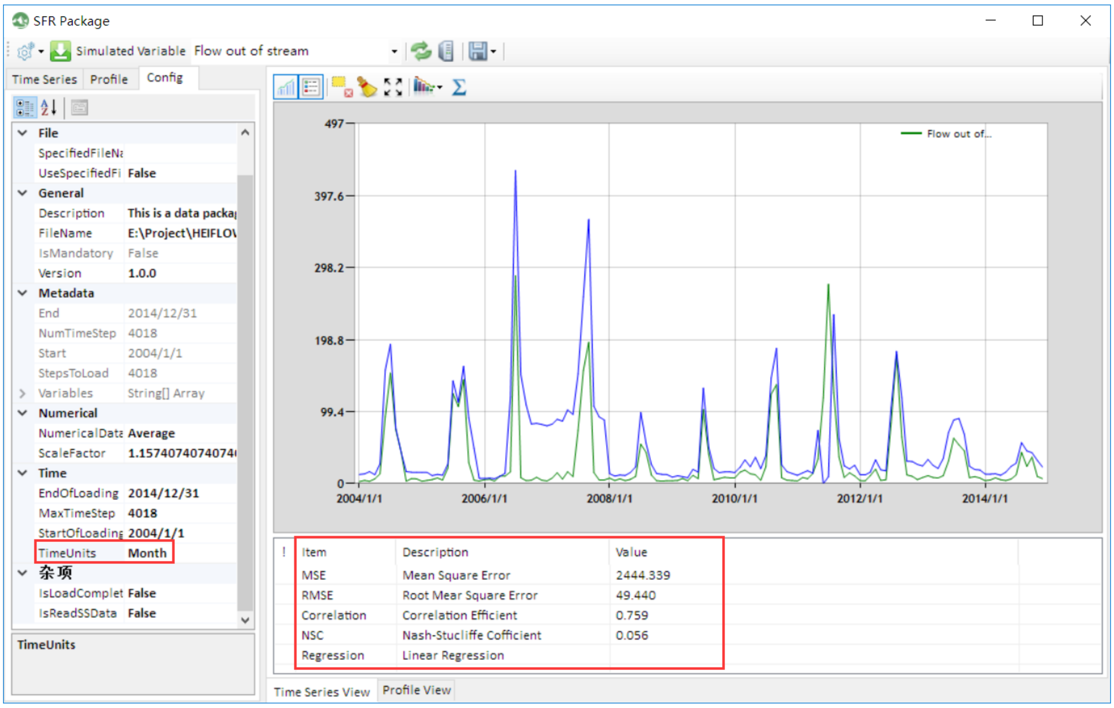

Figure 8-5. Statistics between simulated and observed time series.

Then, we will explore stream-groundwater interactions as follows:

1) Change to Time Series Panel, check the Load complete data, and then click  to load data.

2) Change to Profile Panel, set the Start Segment ID and the End Segment ID as 1 and 35 respectively. Then spatial variation of streamflow along the main stream of the Miho River is presented in the figure (Figure 8-6).

3) Drag the slider bar to view spatial variation of streamflow at different time.

4) Select "Stream loss" in the simulated variables dropdown list and select "35" in the End Segment ID dropdown list to view spatial variation of stream-aquifer interaction flux. The positive values indicate stream leakage to groundwater and positive values indicate groundwater discharge to streams (Figure 8-7).

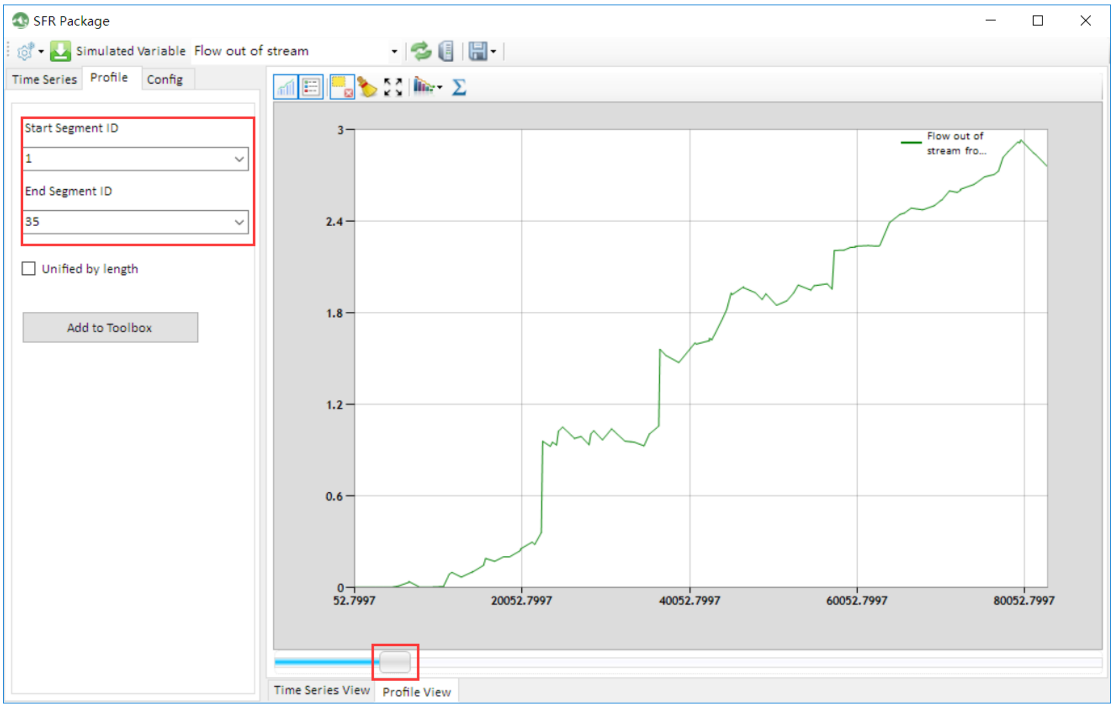

Figure 8-6. Spatial variation of streamflow along the main stream of the Miho River.

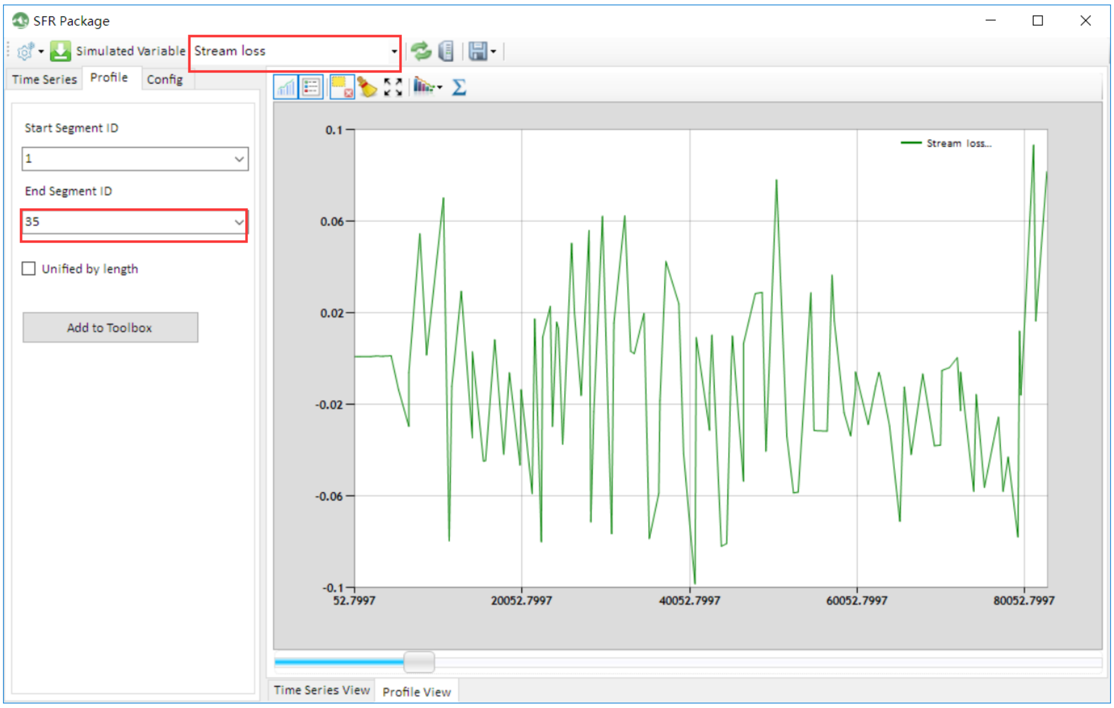

Figure 8-7. Spatial variation of stream-groundwater interaction flux along the main stream of the Miho River.

8.2 Water Budget Analysis

VHF provides a comprehensive water budget analysis tool, which enables to analyze water budget at different scales.

1) Select Model -> Budget Analysis in the Main Menu Bar (Figure 8-8). Click Load button. All budget items are organized into a hierarchy structure.

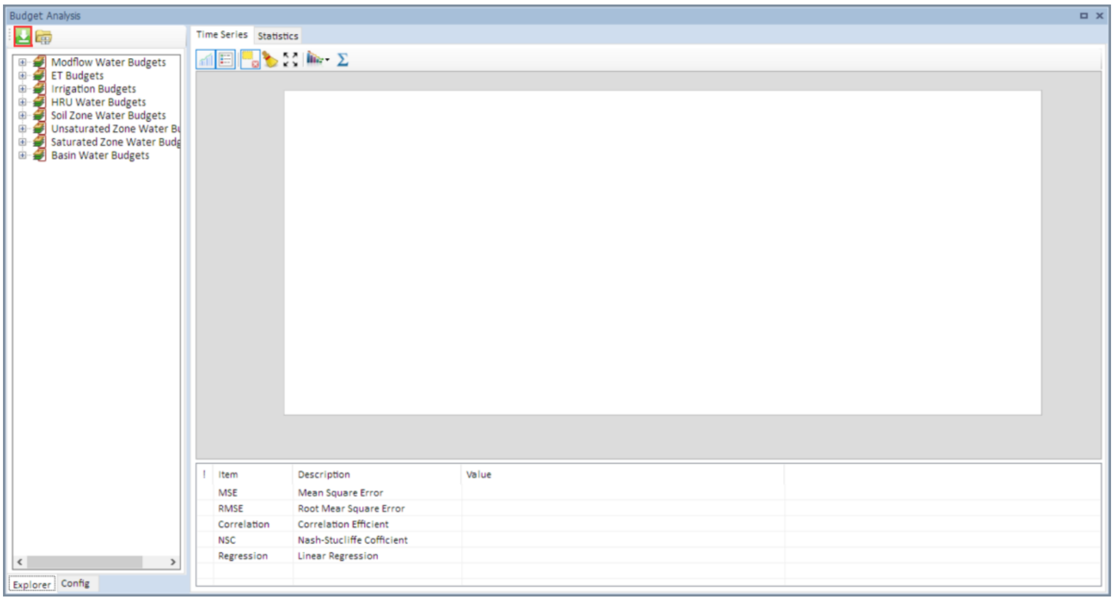

Figure 8-8. Budget Analysis dialog.

2) Double click any budget item to see its temporal variations. As shown in Figure 8-9, the figure shows temporal variation of total ET during the simulation period.

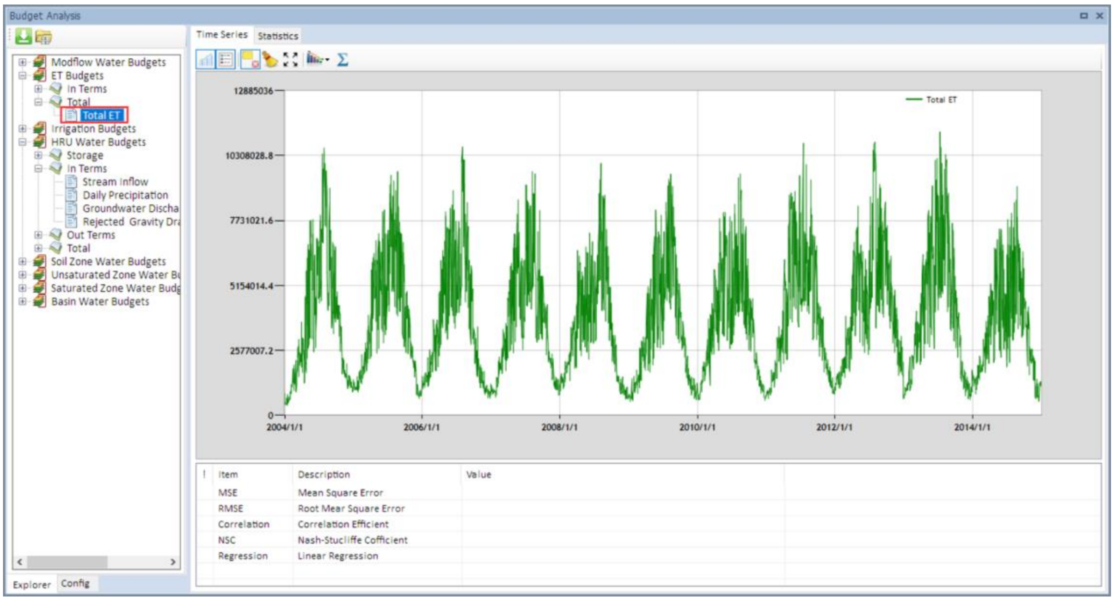

Figure 8-9. Temporal variation of total ET during the simulation period.

3) Right click on “Total ET” item and select Table View, then data values of the total ET will be displayed in the table view (Figure 8-10).

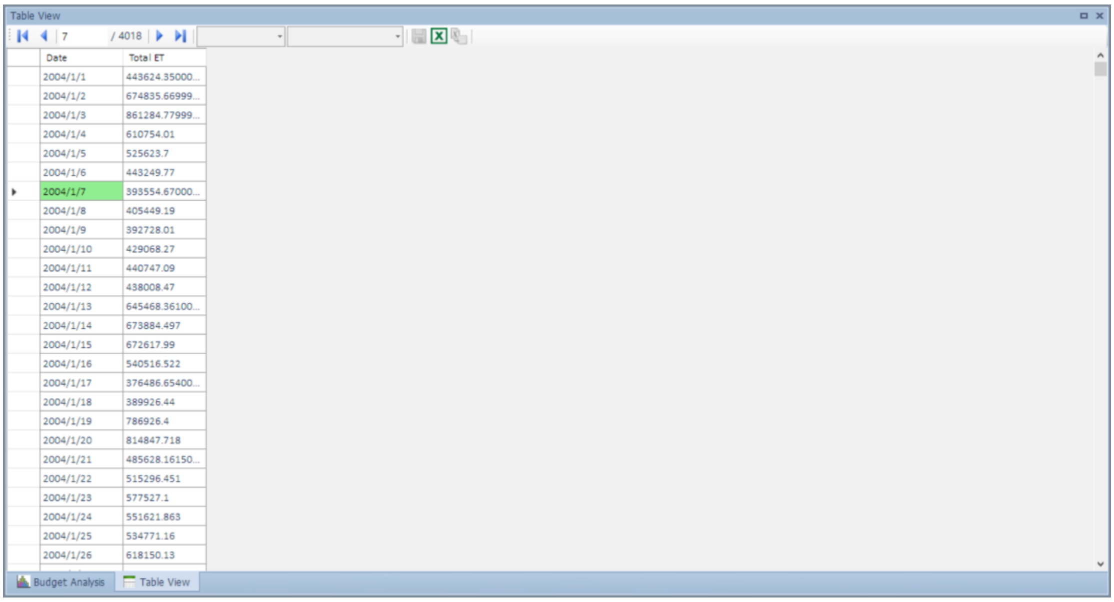

Figure 8-10. Table view of Total ET time series.

4) Right click on “Basin Water Budget” item, and select Flow Budget. Change to Statistics Panel, then the entire region budgets will be presented (Figure 8-11).

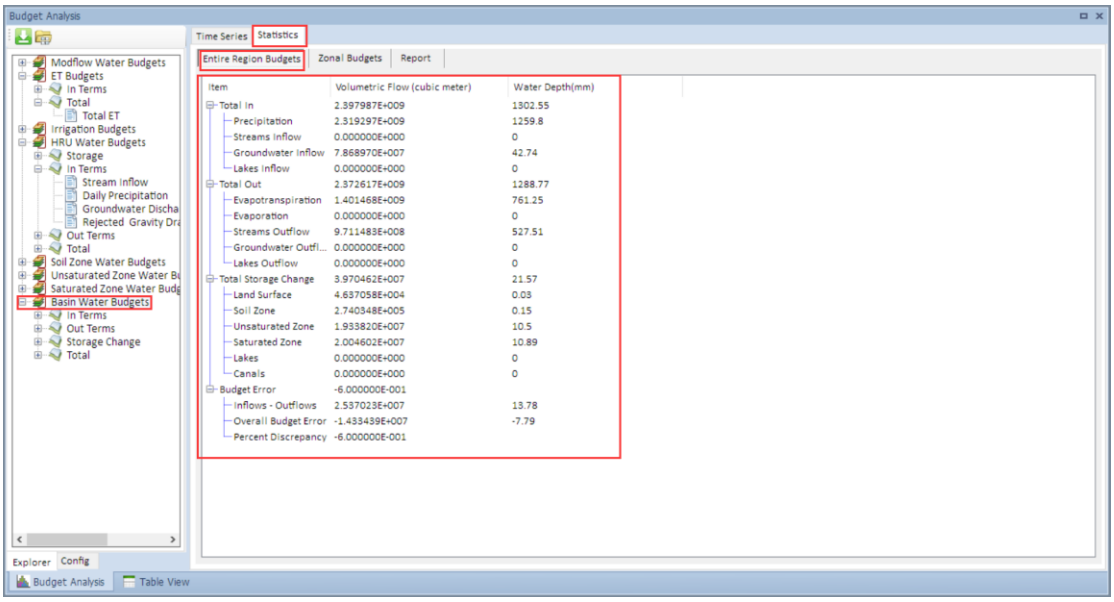

Figure 8-11. Entire region budgets.

5) Right click on “Basin Water Budget” item, and select Zonal Budget. Change to Statistics Panel -> Zonal Budgets. A schematic diagram shows water

budgets in a vertical zone manner (Figure 8-12).

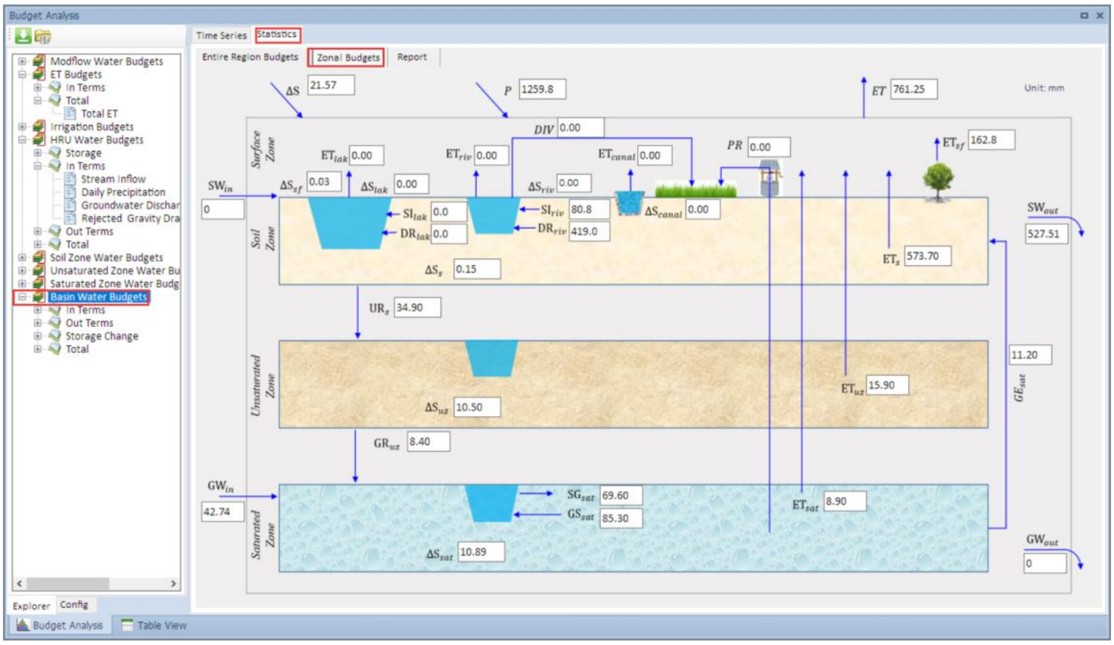

Figure 8-12. Water budgets in a vertical zone manner.

8.3 Animating distributed modeling results

At first, we will probe time series of simulated surface water variables at a user-selected location, e.g., ET, soil moisture, etc.

1) In the Project Explorer, right click on “Animation Output” item, and select Load Variables. Four variables are then loaded as shown in Figure 8-13.

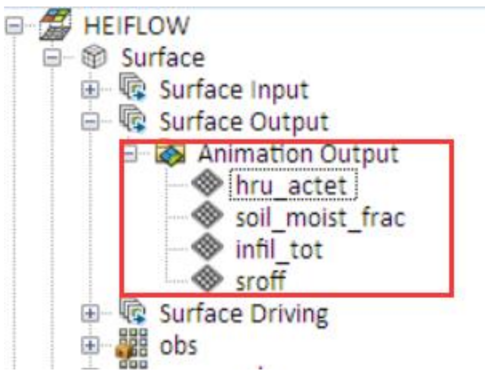

Figure 8-13. Simulated variables contained in the animation output of surface water.

2) Right click on the “hru_actet” item and select Load Data to load actual ET from the output file. When loading, a window is prompted to tell the progress.

3) Right click on the “hru_actet” item and select “Animate…”, then the Animation docking panel appears (Figure 8-14).

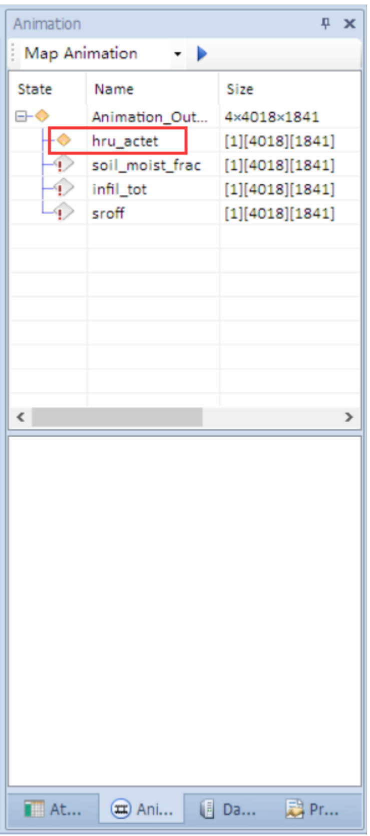

Figure 8-14. Animation docking panel.

4) Left click the “hru_actet” in the above figure. Then the list of date times associated to the hru_actet is displayed in the lower half of the Animation Panel (Figure 8-15).

5) Left click on any item in the date and time list, spatial distribution of actual ET on the clicked date is displayed in the map (Figure 8-16).

6) Select Model -> Active Datasets in the Main Menu Bar, the cursor shape changes to cross, meanwhile the Figure dialog appears. Left click on the map, then simulated time series of actual ET is played in the Figure dialog (Figure 8-17).

7) Close the Figure dialog.

8) You can use similar steps to animate other surface and subsurface modeling results and probe time series at locations of interest.

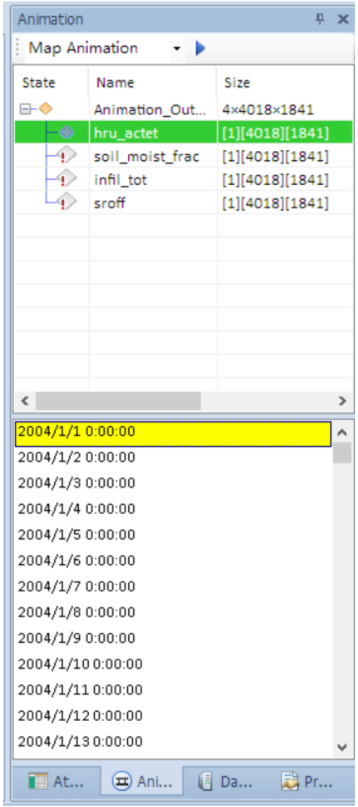

Figure 8-15. The list of date times associated to the hru_actet variable.

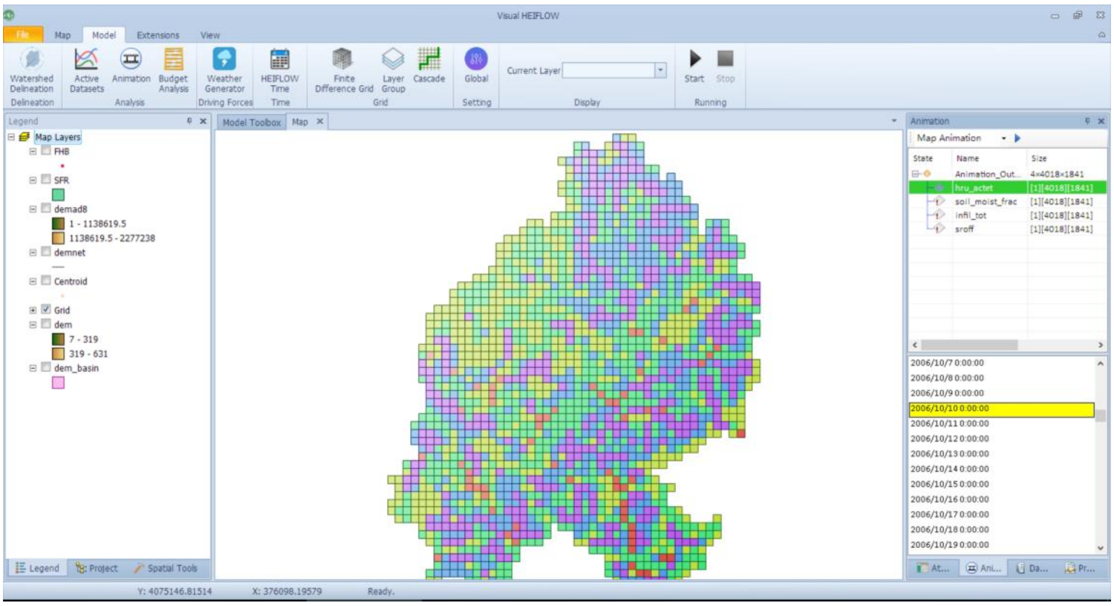

Figure 8-16. Spatial distribution of actual ET on 2006-10-10.

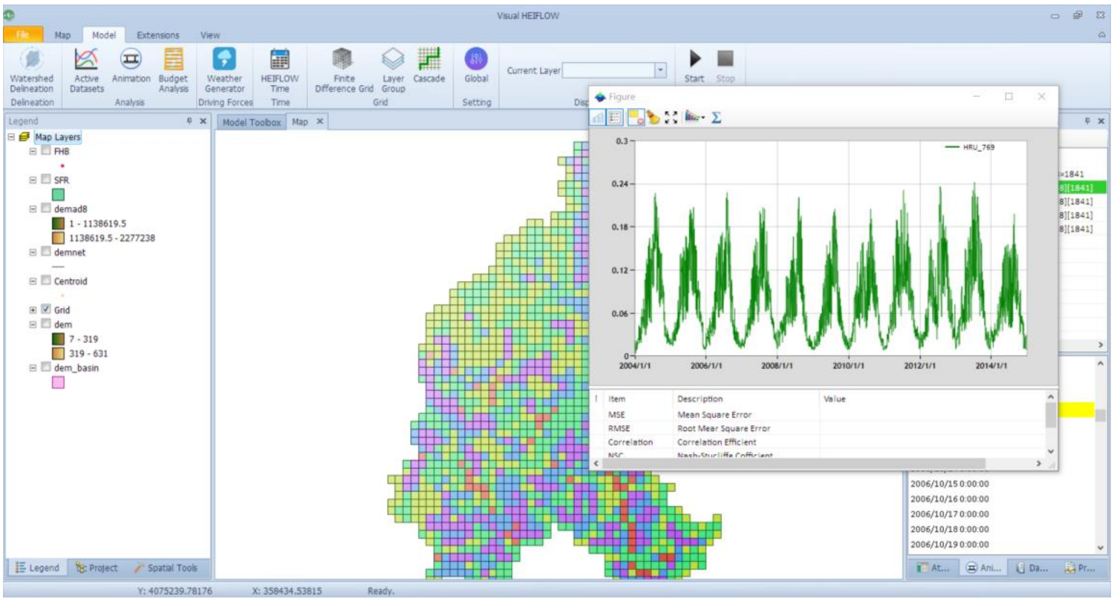

Figure 8-17. Simulate time series of actual ET at the HRU with ID 769.

8.4 Temporal-Spatial Analysis

VHF provides a set of tools to perform temporal-spatial analysis. The following takes Trend Analysis as an example to illustrate this functionality.

1) In the Project Explorer, right click on the “hru_actet” item and select Add to toolbox….

2) In the Model Toolbox floating panel, double click Temporal-Spatial Analysis -> Trend. Set the inputs according to Figure 8-18.

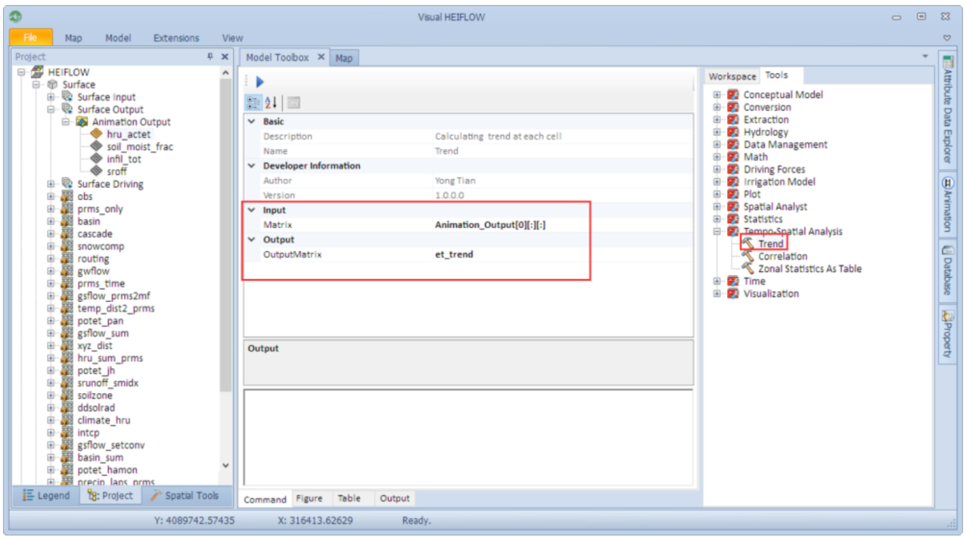

Figure 8-18. Input settings of the Trend tool.

3) A matrix named “et_trend” will be added to the Workspace after running the tool.

4) Double click Visualization -> Grid View, and set inputs according to

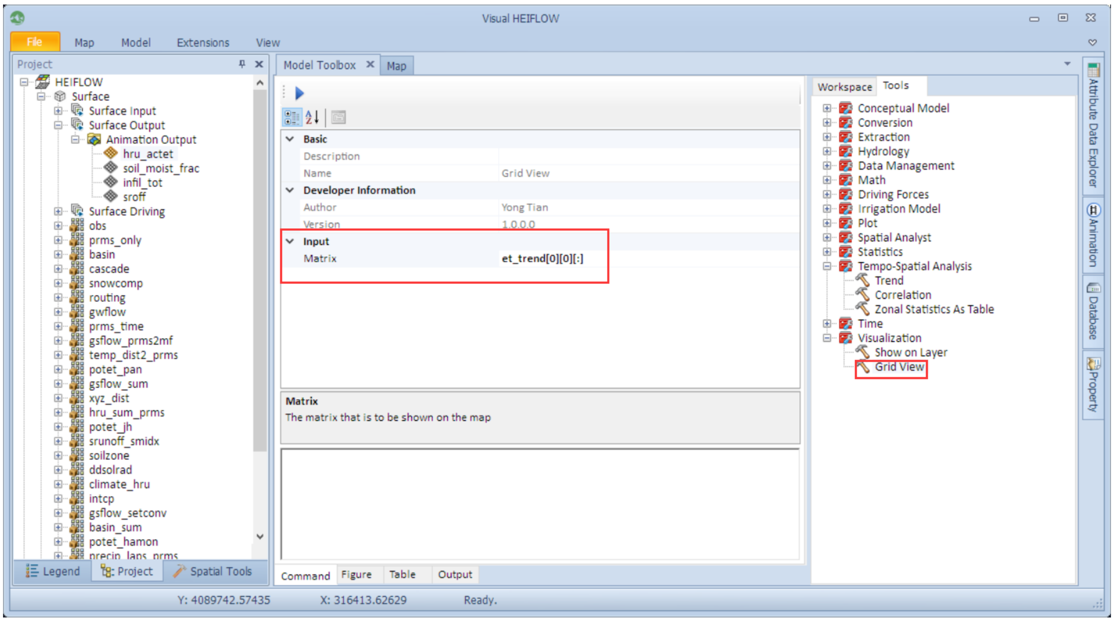

Figure 8-19. Input settings of Grid View tool.

5) Click Run, then the spatial distribution of actual ET trend at daily time scale is shown in the map (Figure 8-20).

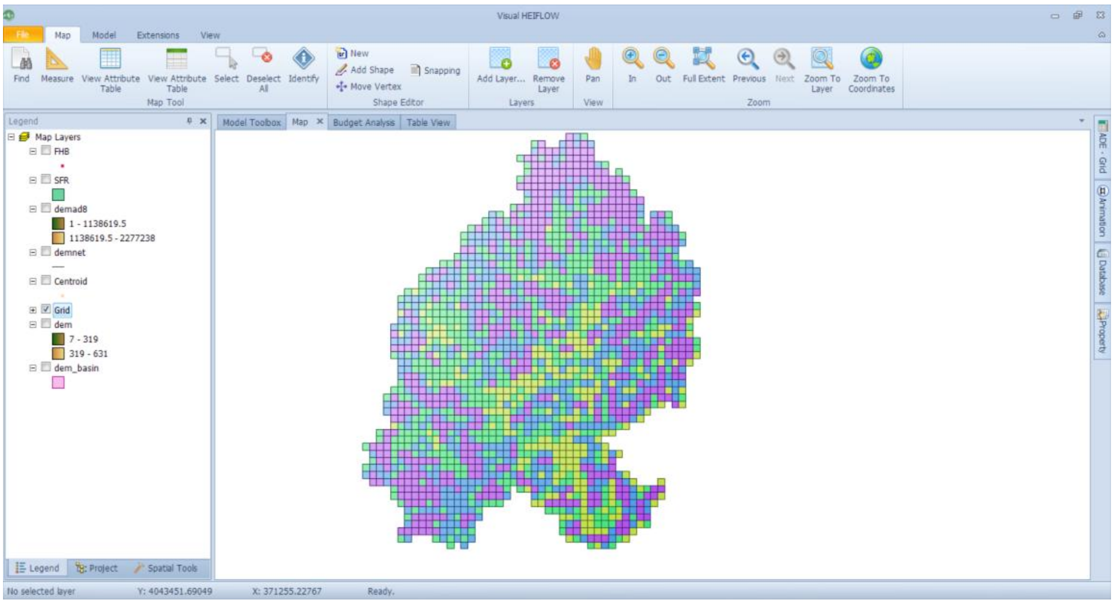

Figure 8-20. Spatial distribution of actual ET trend at daily time scale.